



THE UNIVERSITY
of MANCHESTER

EUROMECH 384
Colloquium on Steady and
Unsteady Separated Flows
Manchester, UK
July 6-9 1998

19981006 037

Book of Abstracts

Supported by
The European Research Office,
United States Army
and
The Centre for Large Scale
Computational Mechanics,
University of Manchester

Unsteady Separated Flows over Manoeuvring Lifting Surfaces

G. Barakos and D. Drikakis¹

UMIST

Department of Mechanical Engineering

PO Box 88, Manchester M60 1QD

United Kingdom

The numerical simulation of unsteady separated turbulent flows around moving lifting surfaces is fueled by the industrial need to analyse and understand flow phenomena associated with the behaviour of aircrafts during manoeuvres. The complex flow phenomena and interactions that occur during super-maneuvrable, high-alpha flight are highly non-linear in nature and include the occurrence of strong suction pressures, rapid movement of the stagnation point, transition of the boundary layer, formation of separation bubbles, production of shocks and most of all generation of an energetic vortical structure, referred to as the *dynamic-stall vortex* (DSV) that, temporarily leads to a significant lift increase (see Figure 1). Numerical simulation of such phenomena appears to be a challenging task. The *phenomenon of dynamic stall* (DS) has been selected as an initial point for the present study. The DS process has been under investigation for about three decades, and significant progress has been made towards understanding of the physical processes associated with rapidly pitching aerofoils beyond the static stall angle of attack. Review of the past work in this area can be found in the papers by Telionis (1977) and McCroskey *et al.* (1991). Computational investigations have been also published more recently (Ekaterinaris (1994)).

Past research has revealed that the accuracy of the numerical calculations is influenced by the accuracy of the turbulence model. Recently, Craft *et al.* (1996) developed low-Re, *non-linear eddy viscosity models* and their studies indicate that these newly developed models are able to give results close to the ones obtained by second-moment closure models.

In the past, researchers have developed Navier-Stokes methods for unsteady inviscid and viscous flows based on explicit schemes, implicit approximate-factorisation schemes or hybrid schemes. The explicit schemes require the use of very small time steps which consequently lead to large computing times while the approximate-factorisation schemes reduce considerably the maximum CFL number especially in three-dimensional flows. On the other hand, implicit unfactored schemes which use Newton sub-iterations provide high CFL numbers and are less sensitive to the choice of the time step than the approximate-factorisation schemes.

In the present work calculations have been carried out, using an implicit-unfactored solver in conjunction with advanced turbulence models, for various flows including quasi-steady flows over aerofoils, impulsively started flow over a NACA 0012 aerofoil, flow around a rapidly pitching NACA 0012 aerofoil, dynamic stall of a harmonically pitching NACA 0012 aerofoil as well as pitching and translating NACA 0012 aerofoil. Using the aforementioned cases, an investigation of the dynamic stall mechanism for the NACA 0012 aerofoil at subsonic flow conditions has also been performed.

¹Author to whom correspondence should be addressed

This study is part of a larger research effort which includes the development and application of advanced turbulence models including non-linear eddy-viscosity models and second-moment closure, as well as, efficient implicit algorithms for unsteady aerodynamic flows and fluid-structure interaction.

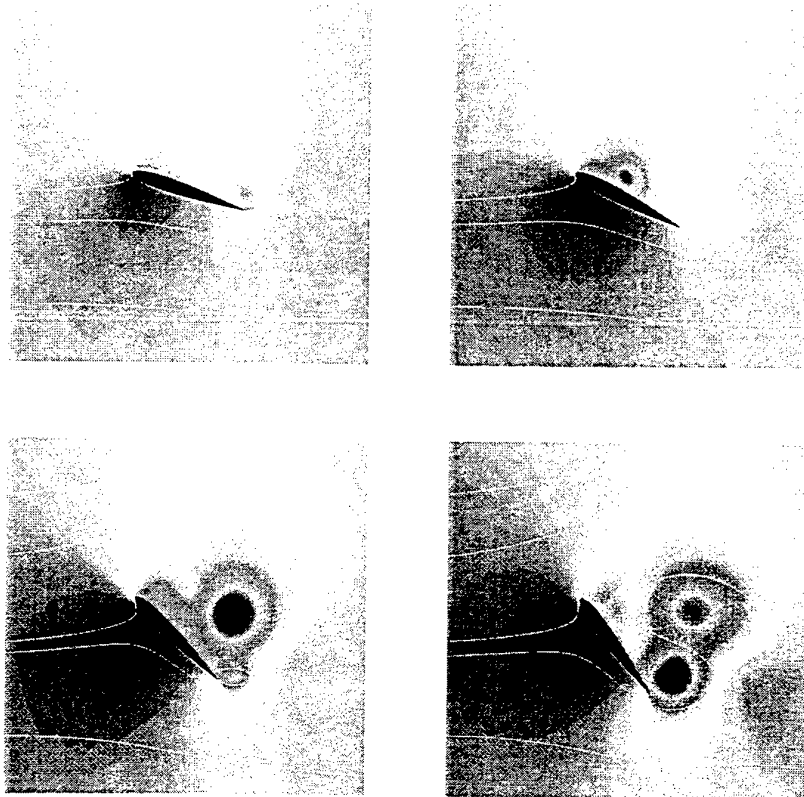


Figure 1: Rapidly pitching NACA-0012 aerofoil ($M = 0.25$, $Re = 10^5$)

References:

- Telionis, D.P., AGARD CP-227, Paper 16, Unsteady Aerodynamics, 1977.
McCroskey, W.J., *NASA TM-81264*, 1991.
Ekaterinaris, J.A. and Menter, F.R., *AIAA J.*, **32**, 2359-2365, 1994.
Craft, T.J., Launder, B.E. and Suga, K., *Int. J. Heat Fluid Flow*, **17**, 108-115, 1996.

COMPUTATION OF SEPARATED COMPRESSIBLE FLOWS USING ADVANCED TURBULENCE CLOSURES

Paul Batten,
Dept. of Mechanical Engineering,
UMIST, Manchester.

This paper focuses on research carried out over the last three years on a British Aerospace funded project to investigate the performance of advanced turbulence models in shock-wave/boundary-layer interactions. The phenomenon of turbulent shock wave/boundary layer interaction has wide-ranging implications for both military and civil aircraft. In situations where the intense adverse pressure gradient across the shock wave is of sufficient strength to induce separation, there is often a strong two-way interaction between the resulting shock pattern and separated flow region. The whole process results in a sharp increase in drag and carries the further risk of large amplitude oscillations.

Recent efforts have focused on turbulence closures capable of providing an adequate description of the Reynolds stresses in a diverse range of flows, which include shock-induced separation, pronounced streamline curvature and strong normal straining. Emphasis has been placed on the derivation of a model variant which is applicable in arbitrarily complex geometries without the use of surface topography-related parameters. The turbulence models considered here include Menter's SST [7], the low-Reynolds number Reynolds-stress models of Jakirlic and Hanjalic [4], Launder and Shima [5], and Craft and Launder [3] and various non-linear eddy-viscosity models [6]. Calculations have been performed using an implicit HLLC scheme developed by Batten, Leschziner & Goldberg [2] in which the convective fluxes determined from the Riemann solver are partly modified to include the effects of the Reynolds stresses.

One example which will be considered is that where a boundary layer meets a fin or a wing, giving rise to a horse-shoe-shaped vortex. In supersonic flow, this interaction is further complicated by the presence of a bow-shock pattern which provokes separation and initiates the formation of the vortex. Numerical calculations have been performed using a variety of models on a Mach 2 flow over a fin/flat-plate junction, studied experimentally by Barberis and Molton [1]. Despite the complicated flow field, the results obtained are in broad agreement with the experimental observations. The SST [7] pressure predictions are in reasonably good agreement with experimental data, however this model fails to predict the multiple vortices observed experimentally. Results from a non-linear eddy-viscosity model [6] also indicated only a single large vortex, probably due to insufficient separation of the normal stresses. A variant of the cubic Reynolds-stress model of Craft and Launder [3] gives similar surface pressure distributions to that of the SST model, but shows much closer agreement with the experimental surface streamline visualisations. Other Reynolds-stress models, such as Jakirlic-Hanjalic [4] and Launder-Shima [5] which both employ a simpler, linear pressure-strain model, also predict multiple vortices but tend to over-estimate the size of the smaller vortex between the reattachment and secondary separation lines, whilst under-estimating the upstream extent of separation.

REFERENCES

1. D. Barberis and P. Molton. *AIAA Paper* 95-0227, Reno NV., 1995.
2. P. Batten, M. A. Leschziner and U. C. Goldberg, *JCP* **137**, 38-78, 1997.
3. T. J. Craft and B. E. Launder, *Proc. 8th Turbulent Shear Flow Symposium*, Munich., 1991.
4. S. Jakirlic and K. Hanjalic. *10th Symposium on Turbulent Shear Flows*, **3**, 1995
5. B. E. Launder and N. Shima. *AIAA J.*, **27**(10), 1319-1325, 1989
6. F-S. Lien, W. L. Chen and M. A. Leschziner, *3rd Symp. on Engineering Turbulence Modelling and Measurements*, Crete, 1996
7. F. R. Menter, *AIAA J.*, **32**(8), 1994

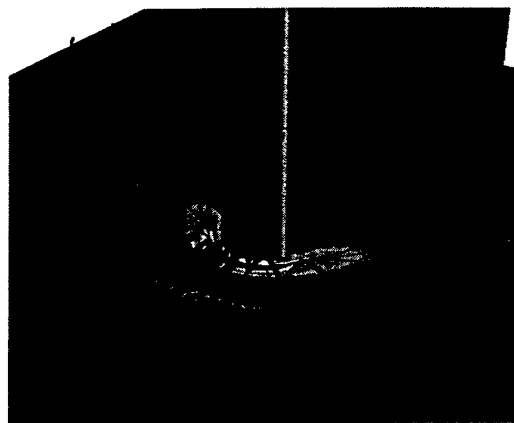


Fig. 1 Overview of flow-field (Reynolds-stress model)

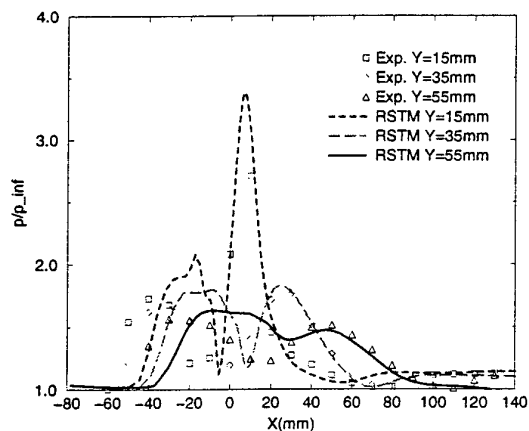
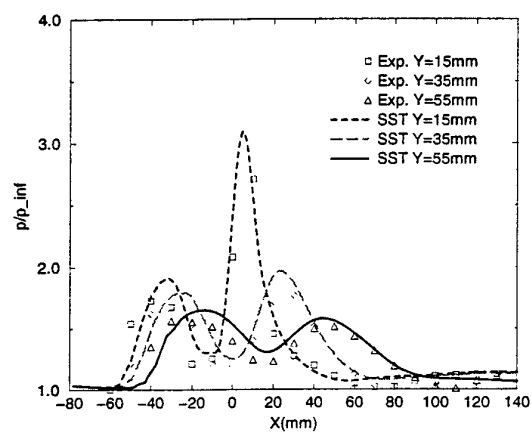


Fig. 2 Plate pressure distributions

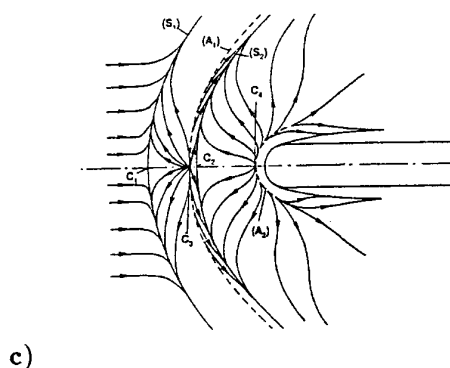
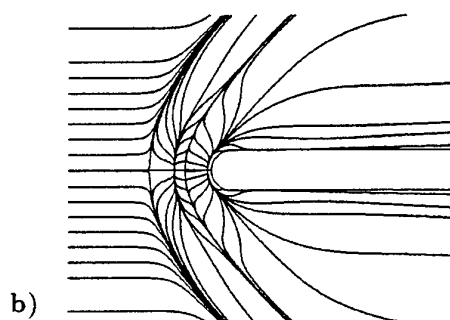
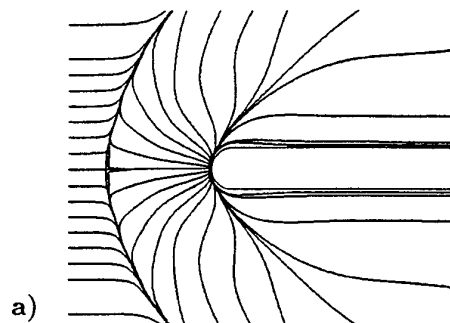


Fig. 3 Surface streamlines : a) SST model
b) Reynolds-stress model
c) Exp. (Barberis & Molton)

Generation of Separation Bubble Through SpinUp in a Liquid Filled Enclosure

Dr.S.Bhattacharyya, Department of Mathematics
Indian Institute of Technology, Kharagpur-721 302, India.
E-Mail : somnath@maths.iitkgp.ernet.in

ABSTRACT :

The present study deals with the spin-up of viscous incompressible fluid confined in a cylindrical enclosure. The initial steady flow due to uniform rotation of the lower end wall with stationary top endwall and side-wall are perturbed either by impulsively co-rotating top end wall with smaller angular velocity or by rotating the container side wall. The transient flow from one steady state to another steady state is studied numerically. We have used a third-order upwind difference scheme in the convective terms of the Navier-Stokes equations for higher values of Reynolds number. The time derivatives are discretised by the ADI scheme. The result shows that a meridional circulation develops in the initial steady flow due to the tilting and stretching of the axial vorticity created by the rotation of the lower end wall. The stream line contours in the vortex core for the initial flow are monotonic for Reynolds number ($Re < 1600$). In the transient flow after certain non-dimensional time a stagnation point appears on the axis of rotation followed by a separation bubble. This phenomena is known as vortex breakdown. As time progresses, the separation bubble enlarges in size and moves downwards. At the ultimate steady state the separation bubble appears closer to the faster rotating lower endwall and the stream surfaces shows a concave shape in the vicinity of the lower Ekman layer. The azimuthal component of vorticity is negative in the core of the vortex except in a region around the axis of rotation where it is positive. The local production of this positive azimuthal vorticity induces a opposite meridional flow which give rises to a separation bubble on the axis of rotation. We found that the increase in rotation ratio of the top-bottom end walls up to certain limit enhances the vortex breakdown. We also find numerically that the impulsive rotation of the container sidewall causes the sidewall boundary layer to separate for a range of Reynolds number and ratios of the side wall to lower endwall rotation ratio.

Separated flows near an upward-facing step

Vladimir V. Bogolepov
Central Aerohydrodynamic Institute (TsAGI)
Zhukovsky-3, Moscow Region, 140160, Russia
Phone: +7-095-5564987, Fax: +7-095-5564337
E-mail: bogol@tsagi.rssi.ru

The flow near an upward-facing step on a flat plate at some distance from its leading edge is investigated. It is supposed the Reynolds number Re_∞ is large but subcritical and the step height h do not surpass the boundary layer thickness $h \ll \delta \sim Re_\infty^{-1/2}$.

Using matched asymptotic expansions method it was shown that for $Re_\infty^{-3/4} \leq h \ll \delta$ the flow near the step is described by Navier-Stokes or Euler equations. The Euler equations solution can be obtained as a limit of the Navier-Stokes equations solution when local Reynolds number Re is large. Calculations results are presented for different local Reynolds number values. It was found the separated region length is comparable with the step height $l \sim h$.

When $h \sim Re_\infty^{-5/8}$ the triple-deck structure of the disturbed region flow near the step is realised and the disturbed region length is $\Delta x \sim Re_\infty^{-3/8}$. It is obvious that separated region with the same length $l \sim Re_\infty^{-3/8}$ can appear upstream the step. Numerical results are presented for the separated flow incipience when external flow is supersonic.

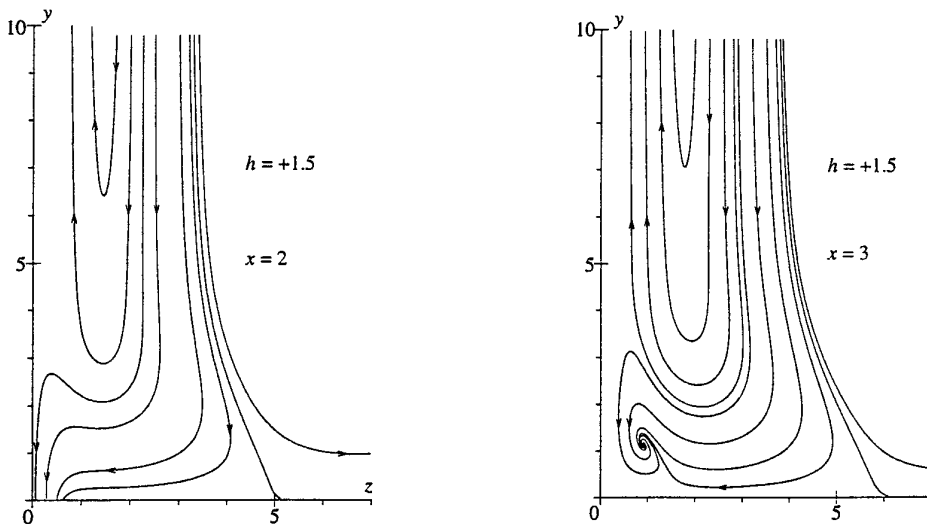
It was shown the separated region upstream the step doesn't exist when the separated region length is of the order of $Re_\infty^{-5/8} \ll l \ll Re_\infty^{-3/8}$.

Numerical Analysis of 3D Boundary-Layer Separation

S. Bos, M. A. Kravtsova & A. I. Ruban

This study is concerned with numerical analysis of three-dimensional boundary-layer separation in hypersonic flow over a 'bell-shaped' obstacle on the surface of a flat plate. The Reynolds number of the flow is assumed to be large, and the triple-deck description of the flow around the obstacle is used. Two numerical methods to solve the viscous-inviscid interaction equations of the triple-deck theory have been employed. The first one is the pseudo-spectral method by Burggraf & Duck. The major advantage of this method is that it effectively treats the interaction equation and allows to solve simultaneously for the entire flow field on each iteration. We found that the method performs well up to a certain value h^* of height h of the obstacle. However, the results prove to be mesh dependent when $h > h^*$.

The second method is based on finite-difference approach and employs an iteration procedure to adjust the viscous and inviscid parts of the flow. Convergence of such iterations is known to be a major problem in viscous-inviscid calculation. To ensure improved stability characteristics of the method, both the momentum equation for the viscous part of the flow and the equation of interaction are treated implicitly. A new numerical procedure has been proposed to solve the resulting algebraic equations on each iteration. The calculations proved to be very stable and mesh independent. For $h < h^*$ they reproduce the results of calculations with the pseudo-spectral method, but show significantly different behaviour of the solution when $h > h^*$. It appeared that perturbations do not decay downstream of the obstacle. Instead two symmetrical vortices are observed amplifying with the distance from the obstacle. Downstream development of one of these vortices is shown in the figure below.



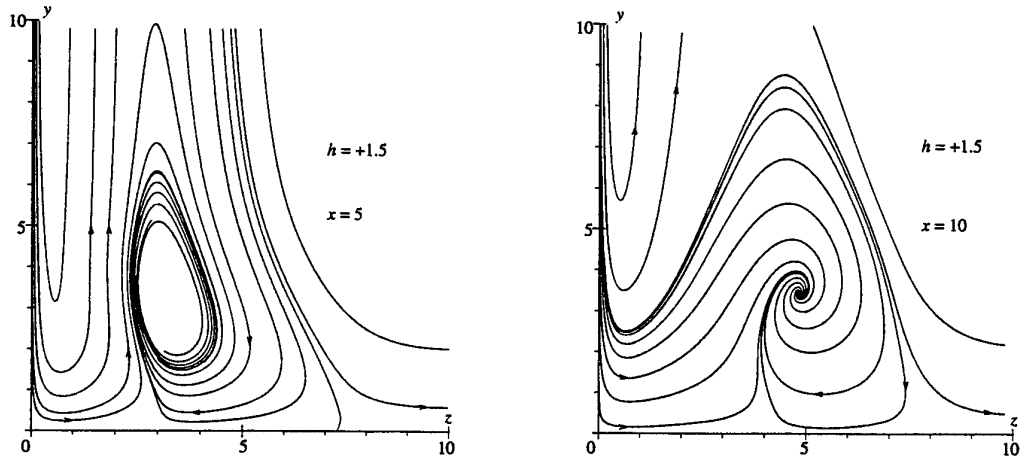


Figure 1: Projection of streamlines upon the yz -plane at different locations downstream the obstacle.

It is clear that such a behaviour can not be reproduced by the pseudo-spectral method which requires all the perturbations to decay with the distance from the obstacle.

S. Bos and A.I. Ruban
University of Manchester
Subsonic and supersonic compression ramp separation

A new 'semi-direct' method to solve viscous-inviscid interaction problems for high Reynolds number separated flows is developed. Both supersonic and subsonic flow separation may be studied using this technique. The method is based upon the vorticity stream function formulation, and is fully implicit with respect to the vorticity equation and 'interaction law'. The interaction law describes the mutual interdependence of the viscous flow near the body surface and the rest of the fluid outside the boundary-layer. The main idea of this approach consists in taking advantage of the particular structure of the viscous-inviscid interaction equations. This allows the entire flow field to be solved simultaneously by using the Thomas Matrix technique. The method proved to be more stable compared to any of the traditional techniques and is also very fast. In this paper the method is used for solving the classical problem of boundary-layer separation in compression ramp flow. Supersonic and subsonic versions of the problem have been studied. In both cases the semi-direct method allows calculation of flow regimes with extended separation regions corresponding to large ramp angles which could not be analysed using other methods.

The nonlinear evolution of a travelling-wave instability in separated flow

R.I. Bowles,
Department of Mathematics,
University College London.

We study the nonlinear development of low-amplitude two-dimensional disturbances in a planar separated flow. The basic velocity profile is taken to be a linear shear above the separated streamline and zero below, to first order, leading to primarily neutral Rayleigh disturbances. However we consider the addition to this basic profile of an arbitrary small correction with curvature thus provoking a small growth rate for a range of wavenumbers. This form of model profile is believed to be applicable to disturbances originating far downstream on the triple-deck scale describing the separation process. Short-scale waves are of a fixed frequency and remain in the vicinity of the original disturbance and their nonlinear development may be studied using techniques developed in Smith, Bowles and Li (1998). Here we concentrate on longer waves which travel downstream at nearly constant speed, with slow modulation through this growth, wave dispersion and an unsteady, viscous, fully nonlinear critical layer. The critical layer is similar to the type studied by Goldstein and Hultgren (1988) with the additional feature of all wavenumbers being present and active. This holds out the prospect of the study of a wave-packet as opposed to the single sinusoidal wave which they consider. Here we do not do this however, preferring to look at periodic disturbances, again with all wave-numbers present, a range of which are unstable. We present computations of the evolution equation for the waves illustrating the growth of span-wise vortices at the critical level. We also investigate the large-time asymptote describing the disturbance growth into a nonlinear, growing, travelling wave, and the variation of the structure of the vortices with amplitude.

Smith, F.T., Bowles, R.G.A. and Li, L. 1998 *Near-wake linear and non-linear stability* (In preparation).

Goldstein, M.E. and Hultgren, L.S. 1988 *Nonlinear spatial evolution of an externally excited instability wave in a free shear layer* J. Fluid Mech. **197**.

Unsteady Vortex-Induced Separation Above a Moving Surface

Professor Kevin W. Cassel
Mechanical, Materials and Aerospace Engineering Department
Illinois Institute of Technology
Chicago, IL 60616

Email: cassel@mmae.iit.edu
Phone: (312) 567-5939
Fax: (312) 567-7230

Numerical solutions of the unsteady boundary-layer and Navier-Stokes equations are considered for the flow induced by a thick-core vortex above a moving surface. Results have been obtained for various wall speeds and several Reynolds numbers in order to determine the influence of the moving wall on the unsteady separation process. It has been shown by Degani, Smith and Walker (unpublished) that unsteady separation is delayed, and eventually suppressed, with increasing wall speed. They obtained numerical solutions of the unsteady boundary-layer equations in Lagrangian coordinates for both the impulsively-started circular cylinder and a rectilinear vortex convecting above a flat surface. It was found that unsteady separation is suppressed in both cases when the wall speed approaches the local external velocity at the outer edge of the boundary layer.

Here solutions of the unsteady boundary-layer equations were obtained in a similar manner for a thick-core vortex in order to corroborate these findings, and it was found that, just as for the circular cylinder and the rectilinear vortex, unsteady separation is suppressed when the difference between the local mainstream velocity and the speed of the singularity goes to zero, *i.e.* $U_e(x_s) - u_s \rightarrow 0$. Numerical solutions of the Navier-Stokes equations were also obtained for the thick-core vortex to determine the effect of finite Reynolds number on this possible control mechanism. It was found that the critical wall speed at which unsteady separation is suppressed decreases with decreasing Reynolds numbers. This is apparently due to the fact that as the Reynolds number is reduced, the streamwise extent of the erupting spike becomes larger and the unsteady separation process is somewhat weaker.

Experimental study of a flow in a vortex cell.

S. I. Chernyshenko, S. V. Guvernyuk, and M. A. Zubin

Institute of Mechanics of the Moscow University

Abstract

Vortices trapped near an aerofoil (that is not shed downstream) can enhance the performance characteristics (see the review in [1]). A vortex cell is a cavity in the aerofoil surface containing such a trapped vortex. Consider (in the case of 2D flow) an aerofoil at a slightly supercritical angle of attack. A small, almost circular cavity can be made in the aerofoil surface upstream of the separation point. Then the velocity profile in the boundary layer becomes more filled in the mixing layer above the cavity. This will prevent massive separation if the angle of incidence is just above its critical value. If the mixing layer above the cavity is located on that portion of the aerofoil surface where the pressure gradient is unfavorable, this can also prevent separation because the dividing streamline may effectively play a role of an downstream-moving wall. The idea of a vortex cell was successfully implemented in the model aircraft EKIP designed by L. Shchukin [2]

The high-Reynolds-number asymptotics of laminar flows of this type is described in [3]. It is possible formally to construct a theory for turbulent flows of this type by replacing laminar boundary layer in [3] with a turbulent boundary layer.

The series of experiments planned and being performed at the Institute of Mechanics aims at determining the conditions when the large-scale vortex remains trapped, studying the structure of the flow in the vortex cell, and estimating which of the two mechanisms described above can be more efficient.

Experiments were performed for a circular cavity with the 60° opening made in a flat wall. The boundary layer upstream of the cavity was fully turbulent. It is found that in this case the vortex remains trapped. The flow structure seems to exhibit a near-wall cyclic turbulent boundary layer. The dependence of the rotation velocity in the cell on the thickness of the oncoming boundary layer was also studied.

1. J. M. Wu and J. Z. Wu, "Vortex lift at a very high angle of attack with massively separated unsteady flow," in: *Fluid Dynamic of High Angle of Attack. IUTAM Symp. Tokyo, Japan, September 13-17, 1992* (ed. R. Kawamura and Y. Aihara), p. 34. Springer (1992).

2. A. I. Savitsky *et al.*, "Method for control of the boundary layer on the aerodynamic surface of an aircraft, and the aircraft provided with the boundary layer control system," United States Patent No. 5417391, May 23, 1995.

3. A. V. Bunyakin, S. I. Chernyshenko, and G. Yu. Stepanov, "High-Reynolds-number Batchelor-model asymptotics of a flow past an aerofoil with a vortex trapped in a cavity," *J. Fluid Mech.*, **358**, 283-297 (1998).

MODELLING OF FLOW AROUND A CYLINDER IN OSCILLATORY MOTION INCLUDING WAVE EFFECTS

Researcher: A.M. Cobbin*

Supervisors: P.K. Stansby* & M.A. Leschziner

Sponsor: EPSRC

(* School of Engineering, University of Manchester)

1. INTRODUCTION

This summary relates to a collaborative project between the Civil Engineering Group at the University of Manchester and UMIST's CFD Group, concerned with modelling the interaction between wave-induced motion and cylindrical bodies which support marine-based oil/gas production systems. Unsteady flow separation and the associated viscous-damping effects on the unsteady forces are of particular interest in this context. A realistic representation of this interaction requires a combination of accurate numerical techniques for resolving the waves and associated sub-surface motion, and a transport model of turbulence which resolves separation and is sensitive to the unsteadiness of the phase-averaged motion.

Previous hydrodynamic analysis of wave-cylinder interaction was based on potential-flow theory with viscous effects incorporated through empirically-based drag coefficients. The aim of the present study is to develop a general RANS method which incorporates wave interaction with cylindrical bodies.

2. COMPUTATIONAL APPROACH

The main building block of the present study is the finite-volume RANS solver STREAM of Lien & Leschziner [1]. This is a general non-orthogonal, multi-block, collocated-mesh, pressure-correction scheme in which convection is represented by van Leer's second order TVD scheme. Implicit time-stepping is effected within an iterative scheme. To improve accuracy and reduce numerical dissipation, second-order time-stepping was incorporated through the use of Gear's method. The low-Reynolds-number $k-\varepsilon$ model of Launder & Sharma [2] was implemented as a base-line model. The exploration of non-linear variants is planned.

To accommodate wave motion, an adaptive-grid Eulerian scheme is implemented in which a grid boundary follows the wave-surface, and the wave-boundary conditions are defined without spatial interpolation. This movement of the mesh requires a re-formulation of the conservation equations and also the introduction of the space-conservation law [3]. The kinematic condition is satisfied by the Lagrangian movement of surface-wave-mesh vertices. A novel feature is a Lagrangian formulation for advancing the velocities of the surface-wave-mesh vertices [4].

The ultimate objective of this study is to develop a general RANS scheme for the prediction of hydrodynamic flows incorporating wave interaction with a body. This is approached by initially examining, in isolation, the wave-induced internal flow around a body, and the wave motion itself. The internal flow is considered by analysing planar oscillatory flow past a circular cylinder, whilst the ability of code to model wave motion is considered by analysing the propagation of a solitary wave. The final stage will be to combine these two capabilities within a general scheme.

3. APPLICATION

The planar oscillatory flow past a circular cylinder is usually defined in terms of the Keulegan-Carpenter number ($K = U_0 T / D$) and the Stokes parameter ($\beta = D^2 / \nu T$), where $Re = K\beta$. Of particular interest are the coefficients of drag and inertia (C_d and C_m). The drag coefficient is of main concern since it determines hydrodynamic damping. The streamline patterns at $\beta = 483$ and $K = 2$ are shown at ten time steps during a half-cycle in Fig. 1. This clearly shows the complex nature of the oscillatory flow with the downstream separation bubble increasing in size over the duration of the half-cycle, and also the creation of an upstream separation bubble. Computed results for C_d and C_m are plotted against K for $\beta = 11240$ in Fig. 2, together with the experimental measurements of Sarpkaya [5] and Marthinsen [6] and the analytic theory of Stokes-Wang. At $K \approx 0.6$ there is a sudden deviation from the (laminar) theory of Stokes-Wang as a result of the transition to turbulence.

This is very close to Hall's [7] theoretical prediction for the onset of Honji instabilities which lead to the onset of turbulence. Beyond this transitional region there is moderate agreement with the measurements of Sarpkaya. Marthinsen's measurements are at variance with the computational results and measurements of Sarpkaya. The computed values of C_m are also in agreement with theory and with Sarpkaya's experimental values. It is expected that non-linear $k-\varepsilon$ turbulence modelling or second moment closure would improve the predictions, especially around the transition range.

Fig. 3 shows the propagation of a solitary wave in a long, straight water channel. For the inviscid case, the wave should propagate without change in shape with any damping indicating numerical error. Indeed, only a very slight negative damping was found. For the viscous case, wave damping was found to be in reasonable agreement with linear perturbation theory. The run-up and reflection of a solitary wave at a vertical wall was also considered and found to produce results in accordance with experiment and theory.

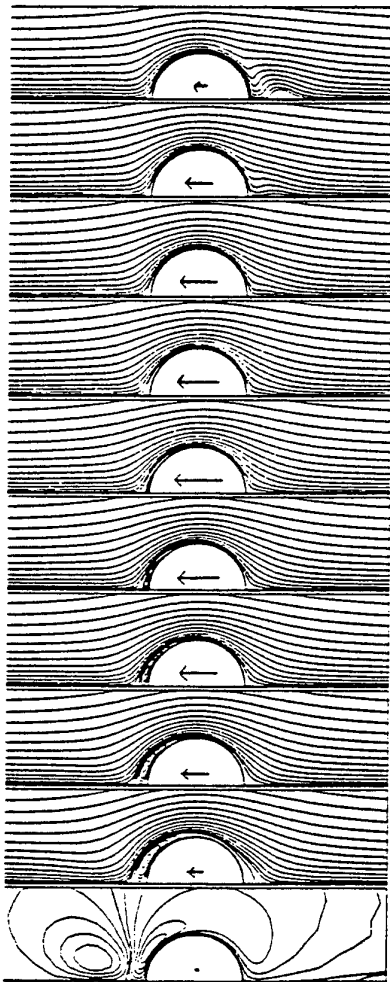


Fig. 1. Planar oscillatory flow past a cylinder (arrow indicates flow direction and magnitude).

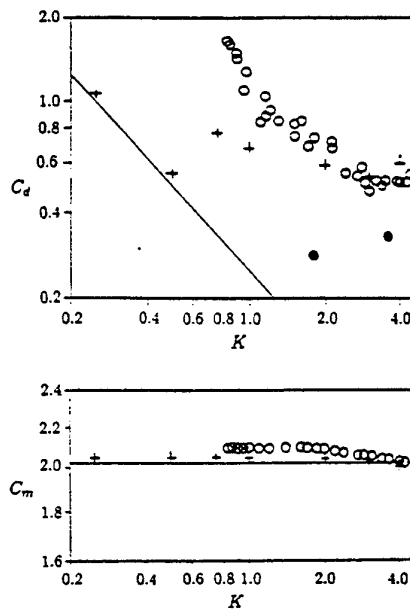


Fig. 2. Drag (C_d) and inertia (C_m) coefficients. —, Analyt.; \circ , expt. [Sarpkaya]; \bullet , expt. [Marthinsen]; +, RANS solver.

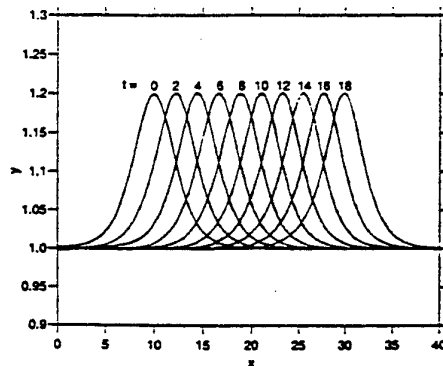


Fig. 3. Computed propagation of a solitary wave.

REFERENCES

1. Lien, F.S. & Leschziner, M.A., *Comp. Meth. App. Mech. Eng.*, 114, p. 123, (1994).
2. Launder, B.E. & Sharma, B.I., *Letts. Heat Mass Trans.*, 1, p.131, (1974).
3. Demirdžić, I. & Perić, M., *Int. J. Numer. Methods Fluids*, 10, p. 771, (1990).
4. Cobbin, A.M., Stansby, P.K., Leschziner, M.A., & Lien, F.S., *Hydroinformatics Conf.*, Copen., (1998).
5. Sarpkaya, T., *J. Fluid Mech.*, 165, p. 61, (1986).
6. Marthinsen, T., *8th Int. Conf. on Offshore Mech. and Polar Eng.*, The Hague, (1989).
7. Hall, P., *J. Fluid Mech.*, 146, p. 347, (1984).

Spatio-temporal structural changes of separated vortex flows

Uwe Ch. Dallmann

DLR – Institute of Fluid Mechanics

Bunsenstraße 10, D-37073 Göttingen, Germany

Uwe.Dallmann@dlr.de

tel: +49 551 709 2442, fax: +49 551 709 2404

Abstract

Changes in laminar boundary-layer flows have been considered for a long time and from different points of views with different physical-mathematical models, ranging from asymptotic analyses to direct numerical simulations, having been adopted. The focus in many studies has been on the region of two- and/or three-dimensional flow separation. Separating boundary layers may exhibit a singularity in finite time (Van Dommelen & Shen (1980)) and high-frequency instabilities (Brown, Cheng & Smith (1988)) and (Cassel, Smith & Walker (1995)) or multiple steady solutions (Gittler & Kluwick (1986), Sommer (1992)). We consider flows with separating boundary layers from a very different point of view. Incipient flow changes rather than (only) incipient changes at flow separations are within our focus.

Spatio-temporal structural changes, i.e. topological changes within the fields of dependent variables have been identified and analyzed. Two-dimensional bubbles and elementary three-dimensional separating flow structures with and without "vortices", with and without reattachment, have been described (Dallmann (1983)- (1998)) and identified in experimental and numerical simulations. We proposed that a two-dimensional separation bubble should be subject to local and global structural bifurcations and a simple bubble should change into a multiple structured bubble before unsteadiness sets in in the vicinity of a point of reattachment rather than close to one of separation (Dallmann (1985)). This has been confirmed numerically by Gruber (1987), Pauley, Main & Reynolds (1989), Rist & Maucher (1994)-(1997) and Herberg & Dallmann (1995). Numerical simulations of different disturbances interacting with a laminar separation bubble confirm a strong amplification of upstream introduced disturbances. Nevertheless, the separation line remains almost unaffected. Unsteadiness and/or laminar-turbulent transition appears first in the vicinity of reattachment.

Our team looks at various other possible temporal and spatial instabilities of a separation bubble created behind a backward facing and smoothly rounded step as well as separated vortex flows around generic configurations like spheres and prolate spheroids. Our objectives are a better understanding of the physical origin(s) of spatial and/or temporal separation bubble instabilities, of the formation and changes of three-dimensional separated flow structures under two-dimensional boundary conditions and of separation bubble transition scenarios.

Results of numerical simulations utilizing a time-accurate numerical method, of new instability analyses and water tunnel experiments will be discussed. The smooth step was chosen for the investigation of the long bubble - short bubble transition phenomenon in flows around airfoils. Starting from low Reynolds numbers, the structural flow behaviour of the bubble with increasing Reynolds number up to and beyond the onset of unsteadiness is investigated. After passing to the unsteady flow regime the changes in the topological, spatial and energetic structure as well as in the global and local temporal behaviour of the flow with increasing Reynolds number are investigated. This is done by Proper Orthogonal Decomposition of the unsteady fields involved. The possibility of investigating the grid dependence of time- dependent numerical solutions via the Proper Orthogonal Decomposition is demonstrated.

Unseparated and separated viscous flow past
an inclined elliptic cylinder.

S.C.R. Dennis
Department of Applied Mathematics
University of Western Ontario
London, Canada N6A 5B7

P.J.S. Young
Department of National Defence
Ottawa, Canada.

ABSTRACT

In this presentation we reconsider a problem to which we have previously given some attention [Dennis and Young (1989,1990)]. In Dennis and Young (1989) some results of numerical calculations of both unseparated and separated steady flow of a viscous, incompressible, fluid past an elliptic cylinder placed asymmetrically to the incident stream were given. However, the details were very sparse owing to space limitations and many of the interesting features of the flow such as the details of the separation and the occurrence of vortices within the fluid have not yet been published. We shall give as much detail as possible for a complete range of inclinations of the cylinder. The flow is assumed to be two-dimensional and to be governed by the Navier-Stokes equations for incompressible fluids.

In Dennis and Young (1990) an attempt was made to relate the asymptotic nature of asymmetric flow at large distances from a cylinder with the circulation around it. However, no precise conclusions were drawn and the question was left open. We have considered this question further since then and some more precise results will be given.

REFERENCES

- S.C.R.Dennis and P.J.S.Young. "Steady viscous flow past an elliptic cylinder at various angles of incidence", Proc. 12th Canadian Congress of Applied Mechanics Vol 2 (1989) 616-617.
- S.C.R.Dennis and P.J.S.Young, "Navier-Stokes flows with circulation" in "Some unanswered questions in fluid mechanics" Appl. Mech. Reviews 43 (1990) 153-170.

Three-dimensional alternatives to the Falkner-Skan family of solutions

Peter W. Duck, S.R. Stow
Department of Mathematics,
University of Manchester
and

M.R. Dhanak
Department of Ocean Engineering,
Florida Atlantic University

We first consider similarity-type boundary-layer flows over corrugated surfaces, the transverse and lateral dimensions of the corrugation being comparable to the thickness of the boundary layer, whilst the streamwise scale of the surface perturbations is taken to be comparable to the streamwise scale of the flow. The governing equations are then similar to those studied in the context of corner boundary-layer flows by Dhanak & Duck (1997). As in this previous work, a thorough understanding of the far-field conditions is necessary, and this leads us to investigate a system of equations analogous to the Falkner-Skan-type, but which allows for a crossflow component of velocity, linearly accelerating in the crossflow direction. It is found that even when the freestream crossflow velocity is zero, in addition to the classical Falkner-Skan family of solutions, another solution family is also present, comprising a jet-like crossflow. When the freestream crossflow is non zero, further degrees of non-uniqueness are found in certain parameter regimes.

Second, an investigation of the streamwise development of the flow (with the similarity constraint removed) leads in many circumstances (including the classical Blasius boundary-layer solution) to leading-edge eigensolutions, which render standard numerical marching procedures ill-posed. Instead, a pseudo-elliptic scheme has been devised, in which both leading-edge and far-downstream conditions are imposed.

**Investigation of the effect of the thin delta wing
cross-section shape on the formation of the secondary
reverse flows in the hypersonic viscous flow.**

G. N. Dudin

Central Aerohydrodynamic Institute (TsAGI)

Zhukovsky-3, Moscow reg., 140160, Russia

Email: yum@tsagi.rssi.ru; Fax: (7095)5564337; Phone: (7095)5564016

Yu. D. Shevelev

Institute for the Computer Aided Design RAN, Moscow, 123056, Russia

Email: shevelev@inapro.msk.su; Phone: (7095)2509280

Abstract

This paper considers hypersonic viscous gas flow over a thin delta wing at zero angle of attack with constant surface temperature. Interaction between the outer inviscid hypersonic flow and the boundary layer may lead to the appearance of strong secondary flows and an essentially nonuniform distribution of the surface heat flux. The resulting secondary flow considerably complicates the pattern of flow past the wing as compared with plane and axisymmetric flows. Under symmetrical flow conditions an increase in the sweepback angle leads to the appearance of secondary reverse flows and a zone of intensified heat transfer and longitudinal friction in the region of the plane of symmetry of the plane delta wing. The region of reverse flow occupies one third of the wing span for sweepback angle near 70 degree. More complex flow in the boundary layer is realized in the case of flow over a delta wing having thickness when the flow is affected both the thickness distribution and the ratio of the characteristic thickness of the wing to the displacement thickness of the boundary layer. It should be noted that in this case regions of secondary reverse flow may be located elsewhere than in the neighborhood of the plane of the symmetry of the wing. As follows from an analysis of received data, the nature of the flow in the boundary layer depends to a large extent on the cross-section shape. The solution of the respective three-dimensional boundary value problem was obtained with the using of the finite-difference method which takes into account the viscous-inviscid interaction phenomena and the local-parabolic direction of three-dimensional boundary-layer equations. In the paper numerical calculations are compared qualitative and quantitative with the experimental data were received by Dudin G. N., Shcherbakov G. I. and others.

Numerical simulation of a ball suspended by an incompressible jet

A. M. Frank

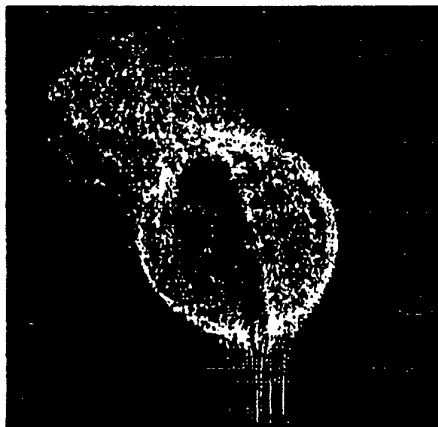
Institute of Computational Modelling, Siberian Branch of RAS,

Krasnoyarsk, 660036, RUSSIA

frank@cc.krascience.rssi.ru, tel: 7-3912-444120, fax: 7-3912-439830

The hydrodynamic effect when a thin upright liquid jet suspends a ball in a presence of gravity force is well known for a long time but still remains unexplained. The full time-dependent problem is rather difficult both for analytical and numerical methods and so there are only some heuristic explanations, based on the analysis of plain steady state jet flow around a fixed cylinder [1]. And the main assumption there was that separation point on the cylinder surface is diametrically opposite to the attachment point. This can be clearly seen in observations and M.A.Lavrent'ev explained it as being caused by viscosity. Assuming this, the nature of retracing force becomes clear - it is just the consequence of momentum conservation, but the stability of the fluid-body system wasn't considered.

Previous 2D calculations [2] showed that this assumption allows to get the stable cylinder motion in time-dependent nonlinear inviscid simulation. They also showed that the viscose stabilization is too slow to be a reason for the assumed (and observed) behaviour of the separation point. There appeared a hypothesis that this mechanism is inertial and essentially three-dimensional. To study it the full 3D simulations have been performed. The new free-lagrange numerical method has been used which belongs to a family of the so-called discrete models of incompressible fluid [3]. This means that the incompressible flow is simulated by a large number of particles subjected by some or the other constraints, corresponding to the incompressibility and boundary conditions, and the governing equations are derived from a variational principle. This allows, in particular, to satisfy the basic mechanical conservation laws for discrete medium, and hence for a numerical model, exactly. These models are especially convenient for simulating free surface flows, fluid-body interactions including.



The inviscid calculations gave stable ball motion in a range of jet to ball radii ratio, density ratio and Froude number. There were also made some experiments and the period of ball oscillations appeared to be in a fairly good agreement with measurements. Calculations confirmed that this effect is fully inertial and allowed to suggest a simple physical explanation which seems to give an example of interesting mechanism of flow separation.

References

1. M.A.Lavrentjev, B.V.Shabat The problems of hydrodynamics and their mathematical models, Nauka, Moscow, 1973, 416p. (in russian)
2. A.M.Frank Numerical simulation of the effect of liquid jet suspending a cylinder, 1997, Doklady Acad. Nauk, vol.357, No.2, 195-198. (in russian)
3. A.M.Frank Discrete models of incompressible fluids, Dr. Phys & Math. thesis, 1993, Krasnoyarsk, 248p. (in russian)

EXPERIMENTAL INVESTIGATION OF SEPARATED FLOWS IN A HYDRODYNAMIC TUNNEL

Gaifullin A.M.*

In a vertical hydrodynamic tunnel two models have been investigated.

The first model represented a low-aspect wing with a parabolic planform and parabolic camber. Such a wing resembles a ski and is called "Nikolsky's ski". According to theory, the main portion of vorticity is generated on the nose part of the wing and propagates downstream as a pair of vortices with cross-sectional dimensions on the order of the curvature radius of the wing's nose. Outside these zones flow can be considered irrotational. The question about a scenario of vorticity generation has remained open.

Separated flow past a wing has been studied experimentally. It turned out that the wing flow is steady-state. Two closed regions of recirculating flow were found on the wing's nose part, which were connected with the wing only at its vertex. Vortex sheets shed from the sharp edges of the nose part are wound on the recirculating regions. The closure of the recirculating regions and further flow occur without noticeable velocity fluctuations.

There was no bursting of the vortex structure either over the wing nor behind it. After cessation of dye injection, the recirculating regions remain visible over some 10 minutes (time of the passage over the wing by a fluid particle is 10 sec, over the nose part - 1 sec).

Interaction between a vortex and a disk-like obstacle placed on the path of the vortex was also investigated. With increasing the size of the obstacle, various regimes of vortex-obstacle interactions were revealed: first, bending around the obstacle; then, the axisymmetric breakdown; and, finally, the spiral breakdown.

The second model consisted of two plates. Between the plates there was a slot whose plane was perpendicular to the trailing edge of the first plate and the leading edge of the second one. The first plate set at zero angle of attack is flat, the second one in the wake of the first plate is bent as an arc. The leading edge of the second plate is shaped to exclude separation. The plates are positioned such that for the streamline shed from the trailing edge of the first plate continues its motion over the convex (with an unfavorable pressure gradient) surface of

*Central Aerohydrodynamic Institute (TsAGI), Zhukovsky-1, 140160, Russia, e-mail: soudakov@glas.apc.org

the second plate. Since the slot between the plates is narrow, the mixing layer behind the first plate forms strata on the boundary layer of the second plate. The purpose of this experiment was to study interaction of the boundary layers of the two plates.

Beginning with some Reynolds number, flow separation was observed on the second plate in an unfavorable gradient zone. It was experimentally found that flow pattern in the neighbourhood of the separation point depends on two Reynolds numbers based on characteristic sizes and velocities of the external flow and the flow within the slot. In accordance with these Reynolds numbers, regimes were observed with one or two separation points spaced closely. As the second regime realizes, a dead-air region forms where fluid velocity is small compared to velocity outside this region.

FLOW STABILITY IN A VORTEX WAKE

Gaifullin A.M.*, Soudakov G.G., Zakharov S.B.

Two causes of vortex wake decay are distinguished. The first of them is associated with the sinuous Crow instability and is observed in the wake behind aircraft having high-aspect-ratio wings. The second cause is the vortex burst. It is mainly inherent in low-aspect-ratio wings. In both the cases characteristic longitudinal dimensions of the wake are far beyond its cross-section size. This fact allows one to describe the wake flow using the well-known thin-body theory. But the first approximation of the theory describes only regular vortex flows. In the present paper higher approximations of the thin-body theory are developed as applied to the description of unstable disturbances in the wake.

Both short-wave solutions (perhaps, corresponding to the vortex burst) and long-wave solutions (sinuous instability) are found. For both the cases the velocity of wave motion is determined in the coordinate systems connected with a fixed observer or an aircraft.

Short-wave oscillations are studied in the framework of a benchmark problem on the evolution of one or two counterrotating hollow vortex tubes. It has turned out that such a flow can be considered as steady in the aircraft-fixed coordinate system. Depending on flow characteristics the regimes are obtained where oscillations amplitudes grow exponentially along wake axis and constant-amplitude regimes. Vortex rotates in a spiral, with oscillation frequency and direction of rotation being associated with flow characteristics.

A program was created to study the long-wave oscillations which makes it possible to compute near and far wakes behind a real aircraft. The lifetime of such a wake is on the order of 1-2 minutes, which corresponds to distances of about 10-20 km from an aircraft. It has turned out that there is no coordinate system where the long-wave wake can be considered as a steady phenomenon. The wake was computed as a surface of tangential velocity discontinuity shed from the wing trailing edge. It was assumed that turbulent cores are located in the vortex rotation centers. The evolution of the velocity tangential discontinuity therewith was computed using the first approximation of the thin-body theory, the turbulent core was computed using the finite-difference method (algebraic anisotropic turbulence model was used to close the system of equations), and the sinuous instability was computed using higher-approximations

*Main-author, Central Aerohydrodynamic Institute (TsAGI), Zhukovsky-1, 140160, Russia, e-mail: soudakov@glas.apc.org

of the thin-body theory. Based on the computed results it was revealed that for concrete aircraft the growing disturbances correspond to a frequency range of oscillations from where the maximally growing one can be found. The sinusoidal oscillation with a specified frequency proceeds in a plane (of its own for each vortex), but this plane slightly changes its inclination as the distance from an aircraft increases.

Nonstationary Separation under Supersonic Flow Deceleration in Ducts

G.F.Glotov, N.V.Gurilyova, M.A.Ivankin, V.V.Bogdanov, B.D.Bryantsev
Central Aerohydrodynamic Institute (TsAGI)
Zhukovsky-3, Moscow Region, 140160, Russia
Fax: +7-095-5564337
E-mail: igor@lipatov.rosmail.com

At present a supersonic flow deceleration in a long duct ("pseudoshock") is studied and calculated within the scope of stationary approaches. Meanwhile, there is a number of works (MacLafferty et al., Ikui et al., Arai et al., Gurylev et al.) where a pulsating character of the flow was shown. Pressure pulsations frequencies ($f_1 = 100 - 300\text{Hz}$), oscillations frequencies of a shock system ($f_2 = 10 - 40\text{Hz}$), mean-square deviations of pressure pulsations on the wall $\sigma_1 = \sigma / (0.7 * Pc * M_c^2) = 3 - 5\%$ were determined. However, in view of the absence of a gasdynamic model of such flow a number of phenomena either was not explained or explanations had a contradictory character. On the basis of experimental investigations of an efflux of a supersonic flow from an axisymmetric duct ($d = 106,7\text{mm}$; $l/d \leq 11.35$; $M_c = 2.34$; $Re = (2 - 4) * 10^7$; off-design factor $n = P_c/P_e \leq 0.6$) and an analysis of literature data G.F.Glotov (the leader of the work) suggested the model of the nonstationary flow, unlike commonly used models of the stationary flow with the pseudoshock. This nonstationary flow model is characterised by longitudinal and lateral oscillations of the whole flow with a low frequency which are caused by a pulsating separation. In the regimes $0.33 < n < 0.5$ in present experiments an alternating formation of a closed/partially-opened circular separated zone at a wall was observed. As a result, the flow executes longitudinal and lateral oscillations over the certain range of amplitudes (the self-excited regime of the flow of the expendable type). For the flow with a developed pseudoshock in the duct characteristic parameters of the pulsating flow: the pressure pulsating frequency ($f_1 = 10 - 270\text{Hz}$) in the separation

zone on the wall, a value of its mean-square deviation ($\sigma_1 = 3 - 4\%$) were determined. A duct length decrease to $l/d \leq 3 - 5$, a duct section shape change affect as well as an increase of n , hereat characteristic properties of the flow are conserved. Experimental investigations of the deceleration of the supersonic flow ($M = 1,8 - 2.5$) in a rectangular flat duct ($hxb = 36 \times 40 \text{ mm}$, $(l_1 = l/b \leq 5)$) showed that the suggested above nonstationary model was verified for a free pseudoshock. For the free pseudoshock there is typical an availability of pulsating unsymmetrical separated zones under a bow shock wave near by as upper and lower as sidelong walls of the duct. As M number rises, the amplitude and frequency of shock oscillations rise. The suggested model offers new opportunities for calculations of pulsating flows in ducts and control of supersonic separated flows.

The work was carried out at the financial support of Russian Foundation of Basic Research (project 97-01-00594).

Artificial separation as a mean for generation of large stationary vortices and decreasing the body hydrodynamic drag

V. Gorban, I. Gorban

Institute of Hydromechanics National Science Academy of Ukraine,

E-mail: vin@ihm.kiev.ua, Phone: (38044)546-42-71

In the present work, the schemes which describe the forming of local separation zones in near-wall flow with the help of special control boundary irregularities are considered. To simulate the flow evolution, the numeral discrete vortex method based on model of ideal incompressible fluid is used. Within the frameworks of two-dimensional problem, dynamic properties of the vortex structures developing in separation zones, their stability and receptivity to external perturbations are investigated. The schemes of near-wall flow with stationary separation zones, which are steady in respect to external perturbations and generate minimum vorticity into a boundary layer, are of most interest because of its practical importance. To extract conditions for existence such zones and research their properties, we use the standing vortex model, in which one vortex simulates all separation zone vorticity. The demand of immovability of the vortex and the Kutta condition satisfied in sharp edges of the flow boundary are used to determine the vortex parameters. The calculations were fulfilled when the vortex was located inside a cross groove, in the area between two curvilinear plates and in the system "control plate - body". The special choice of the irregularity shape was shown to allow to satisfy the Kutta condition in both sharp edges. It means that such vortex is able to remove the generation of vorticity behind the separation zone.

The standing vortex was found to have the characteristic eigenfrequency. As a result, its motion in perturbed flow essentially depends on a frequency of external perturbation. Moreover the interaction between the vortex and the perturbation has the resonant character. If their frequencies are close one to other, the amplitude of perturbed motion of the vortex around its stationary position will increase and the flow pattern will be changed. The developed method allows to estimate optimal geometrical parameters of the boundary irregularity which ensures the forming of steady local separation zone. The derived optimal shapes were tested through discrete vortex model. The favourable choice of control irregularity was obtained to lead to decreasing the separation zone sizes and the intensity of the vortex structures which are thrown into a near-wall flow. Due to these factors, the hydrodynamic drag of body decreases.

Turbulent flow separation - a PIV study

by

Jonas Gustavsson & Barbro Muhammad-Klingmann

Dept. of Mechanics, Royal Institute of Technology, S-100 44 Stockholm, Sweden

A turbulent boundary layer is subjected to adverse pressure gradients of different strengths, and the pressure distribution, wall shear stress, mean and perturbation velocities are studied. Measurements were made using *Particle Imaging Velocimetry* (PIV), and the results are compared to those obtained from other techniques such as static pressure measurements, Preston tubes and hot-wire anemometry. Three different cases are studied, namely (i) a decelerating boundary layer on the verge of separation, (ii) a separation bubble induced by placing a trip in the aforementioned boundary layer, and (iii) a stronger pressure gradient causing global separation. The separation trip also serves the purpose of avoiding instability of the separated region, which tends to move both in the streamwise and the spanwise direction.

The data for the attached boundary layer were compared to hot wire measurements, and different methods of evaluation of the wall shear stress were compared. For cases where the adverse pressure gradient is strong and persistent enough to cause mean flow reversal, some currently used criteria for separation prediction are tested. In the case of the separation bubble, it was found that the shape factor H decreases from its maximum value of around 15 to about 4 at reattachment, defined as the wall position where the back-flow coefficient at the wall decreased below 50%.

An example of the PIV results is shown in figure 1, which is taken at the upstream end of a separation bubble. In the region closest to the plate, the back-flow tends to be much higher than some distance away from the plate, and this poses high requirements on the spatial resolution. At the same time, the range of velocities through the boundary layer is quite large, giving a delicate balance between accuracy and dynamic resolution. These and other aspects of the PIV method will be discussed.

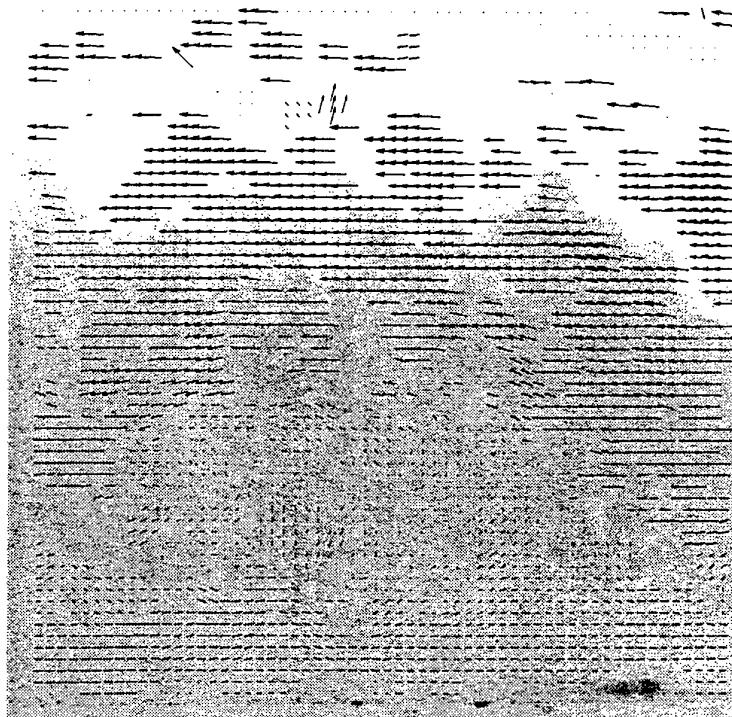


Figure 1. Particle image and velocity field at the beginning of a separation bubble. The free-stream is from right to left, and the position of the wall is at the lower end of the picture.

Experimental Study of a Transitional Two-Dimensional Separation Bubble

HÄGGMARK C. P., BAKCHINOV A. A. & ALFREDSSON P.H.

Dept. Mechanics, Royal Institute of Technology, S-10044 Stockholm, Sweden.

A transitional two-dimensional separation bubble was created on a flat plate in the MTL wind tunnel using an adjustable hump at the upper wall to establish the pressure gradient (figure 1). Linear and nonlinear development of natural and artificially introduced wave packets in the boundary layer were studied using hot wire anemometry. In the vicinity of the source TS-waves typical for the Blasius boundary layer were observed. A third maximum in between the two maxima of the TS-wave amplitude profile was found to be associated with an inflectional streamwise mean velocity profile in the adverse pressure gradient and separation regions. Both velocity correlation measurements and smoke visualization revealed that the unsteady vortical structure was still two-dimensional just prior to the reattachment of the shear layer, whereas both steady and unsteady flow structures in the reattachment region were found to be essentially three-dimensional (figure 2).

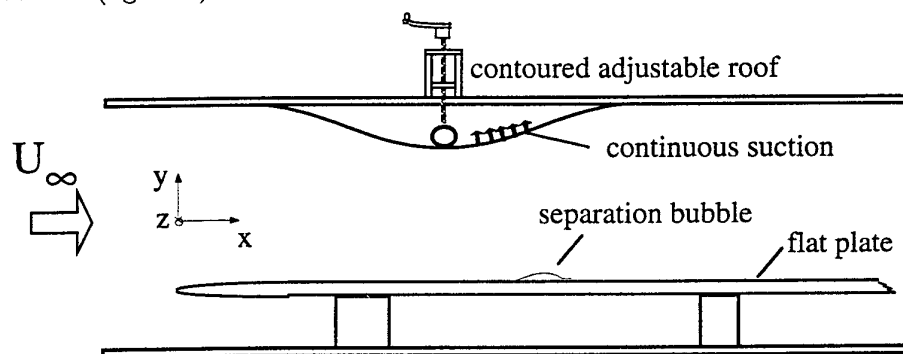


Figure 1. Experimental set-up

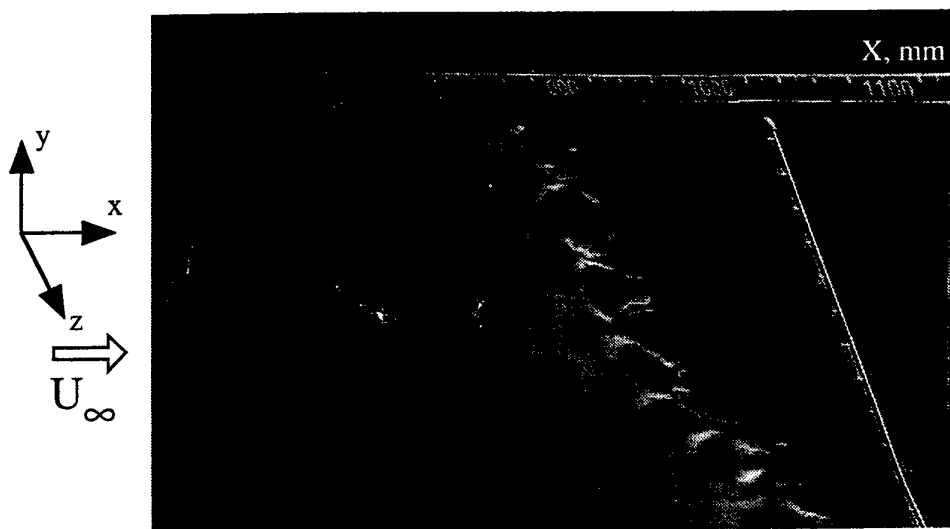


Figure 2. Separation in the presence of a periodical 2D wave, $f^* = 101$ Hz

On the origins of instability and unsteadiness in laminar separation bubbles

Stefan Hein, Vassilios Theofilis & Uwe Ch. Dallmann

DLR – Institute of Fluid Mechanics

Bunsenstraße 10, D-37073 Göttingen, Germany

Stefan.Hein@dlr.de, Vassilios.Theofilis@dlr.de, Uwe.Dallmann@dlr.de

tel: +49 551 709 2442, fax: +49 551 709 2404

Abstract

Experiment and numerical simulations have shown that the formation of a laminar two-dimensional separation bubble within a boundary-layer flow at large Reynolds number will strongly amplify both short- and long-wavelength disturbances leading to transition and turbulent reattachment. The role of three-dimensionality as well as the origin of unsteadiness, however, are far from being understood in a satisfactory manner. Furthermore, unlike the case of attached boundary layers, the strong streamwise variation of the flow prevents uncritical use of available instability analysis tools.

We have successfully applied alternative instability analyses based on the Navier-Stokes equations which, firstly take into account the streamwise gradients of the flow and, secondly treat the limits of small and large wavenumbers independently. Excellent agreement has been obtained in the case of short-wavelength Tollmien-Schlichting type of instability by comparison with available direct numerical simulation results. Long-wavelength disturbances, on the other hand, are being monitored with respect to the onset of unsteadiness and three-dimensionality of the flow. Both linear and nonlinear instability results of both approaches will be presented at the time of the Conference.

HIGH MACH NUMBER SEPARATED FLOWS

Richard Hillier and Andrew Jackson,

Department of Aeronautics, Imperial College of Science, Technology and Medicine,
London.

Separated flows at supersonic/hypersonic speeds have been the subject of intense study over the years. They are of particular importance, for example, in base regions, surface recesses or cavities, and in regions of interaction between shock waves and boundary layers.

This paper comprises two parts. The first will survey some of the relevant work on cavity-induced and wedge- or flare-induced separations, in particular highlighting cases where significant flow unsteadiness is observed. The second will describe some of our own work on cavity flows at hypersonic speeds, an area where unsteadiness is much less pronounced but where other demanding phenomena appear instead.

One objective of our work has been to develop as close an integration as possible between CFD and experiment. The purpose is to achieve several interlinked objectives: the experiments can provide data for CFD evaluation/ validation; CFD can 'probe' regions of the flow field which are inaccessible to experiment; CFD can define regimes, or critical areas, of interest and is therefore an essential and integral part of model design.

Our paper will present both CFD and experiment for two-dimensional (axisymmetric) and three-dimensional separation. In the latter case a model design has been developed which enables us to produce flows with an increasing severity of three-dimensionality and which will therefore provide a good, structured hierarchy of test cases. The results which we will present include:

- A brief description of the numerical modelling and the general approach to experiment design.
- Preliminary CFD studies used to check cavity flow establishment times - a critical factor when using 'short-duration' facilities (such as ours), typical of those used for most high Mach number studies.
- Experimental studies for both fully laminar and fully turbulent flows.
- Extensive accumulation of surface heat transfer data - one of the more demanding quantities for CFD assessment.
- Schlieren visualisation of the free shear layer development and reattachment shock system.

Separation in Valves Governing Steam Turbine Inlet Flows.

P.D. Iredale & S.I. Hogg

Engineering Department
University of Leicester
University Road
Leicester
LE1 7RH.

Abstract

Valve systems used to control the flow of high pressure steam into the cylinders of steam turbine sets usually consist of a stop valve and governing valve combination. The former is used to isolate the steam supply from the turbine, whereas the role of the latter is to throttle the inlet flow for running at part load conditions. Figure 1 shows a typical governing valve geometry.

When partially closed, the operation of the governing valve relies on flow separation off the valve head and seat to generate loss and throttle the flow. The ultimate aim in this type of valve design is to avoid separation and therefore essentially eliminate loss when the valve is fully open, and to have stable and controllable separations at all other valve lifts. The output of steam turbine sets are typically 100's MW and so the inlet valves control extremely high enthalpy steam flows. Any significant unsteadiness in the valve flow can result in unacceptable mechanical vibration of the valve, which in extreme cases can lead to failure.

Results will be presented from work currently being undertaken into valve flow instabilities at Leicester University in collaboration with GEC ALSTHOM Turbine Generators Ltd. At high lifts, the Mach number of the steam flow between the head and the seat is sufficiently low for the flow to be treated as incompressible. Water is used as the working fluid in the tests at Leicester to accurately model the flow in a fifth scale acrylic model valve under high lift conditions. The main advantage of testing in water compared to air or steam is that water lends itself more readily to laser flow visualisation and flow measurement. Results from both laser light sheet visualisations and transient pressure measurements of the valve flow are presented.

Laser light sheet illumination and high speed cine photography have been used to visualise the highly three dimensional valve flow. A range of valve head geometries have been tested. The results of the flow visualisations show the presence of stable and unstable separation zones and their influence on the valve flow. Methods for stabilising the separations by modifying the valve head and seat have also been investigated and results from these tests are described. Figure 2 shows pressure signals in the frequency domain, which were measured in the exit pipe from the valve at the location shown in Figure 1. In one of the tests, the valve head has been modified to stabilise the separation from it and the effect on the unsteadiness of the flow exiting the valve can clearly be seen.

Fig. 1: Typical Governing Valve Geometry.

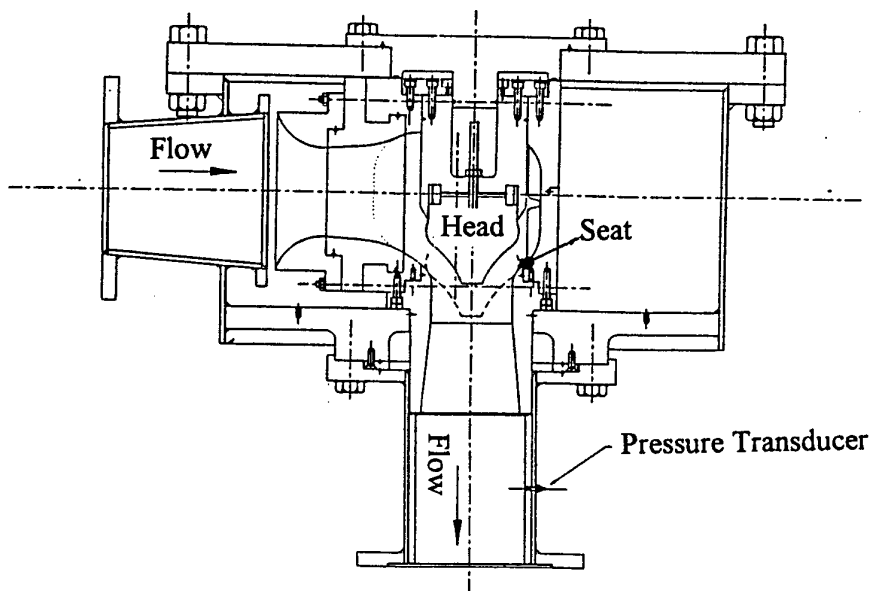
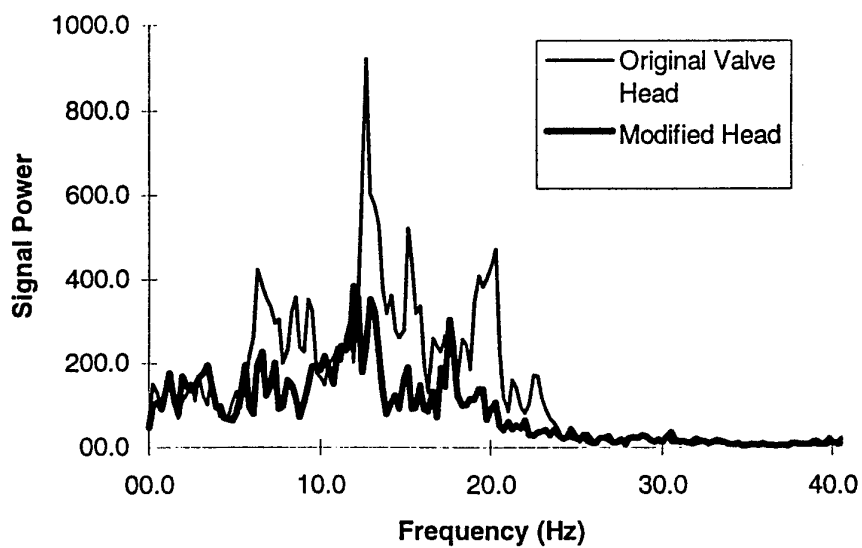


Fig. 2: The Effect of Modifications to the Valve Head on Exit Pipe Pressure Signals.



Multiple hysteresis of the aerodynamic characteristics at low Reynolds numbers

Kabin S.V., Kolin I.V., Soukhanov V.L., Svjatodukh V.K., Shuchovtsov D.V.
TsAGI, Moscow, Russia

It is known that aerodynamic forces and moments acting in wind tunnels on the aerodynamic models, especially on the high aspect ratio wings with large thickness, are highly nonlinear in the range of angles of attack, where flow separation begins. In this case the aerodynamic characteristics are sensitive to the variations of the parameters of the experiment (angles of attack and sideslip, Reynolds number, amplitude and frequency of model oscillation in the case of dynamic tests, etc.). The hysteresis is a well known phenomena for static case and it is investigated intensively for unsteady aerodynamic characteristics too. It is evident that it is highly difficult to develop the adequate mathematical model which describes the aerodynamic coefficients variations and takes into account the unsteady aerodynamic effects in this case.

One of the most interesting problems in this field is the problem of the multiple hysteresis existence. The results of experimental investigations of this phenomena made in low speed wind tunnel at TsAGI using rectangular wing with airfoil NACA 0018 are presented in this paper. It was shown that in some cases not only one, but also two, three, etc. stable hysteresis loops in the dependencies of aerodynamic forces and moments acting on the model can exist. Along with the measurements of total aerodynamic forces and moments acting on the model, the flow pattern on the wing surface was investigated. It was shown, that in the case of multiple hysteresis the existence of stable nonsymmetrical flow structures on the wing is possible, which lead to arising of nonsymmetrical forces and moments for symmetrical (at infinity) flow conditions. A special attention is paid to the investigation of the condition where multiple hysteresis occurs and to the investigation of the stability of the internal branches of the hysteresis loops. Along with the measurements of "static" aerodynamic forces and moments the investigations of aerodynamic damping derivatives with various frequencies and amplitudes were made using forced oscillation test rig. A correlation between "static" and "dynamic" hysteresis of the aerodynamic characteristics was stated. Some problems of the methodology of experimental investigation in "static" and "dynamic" cases are discussed.

E-mail: FLIGHT@TSAGI.RSSI.RU

FAX: (095) 556-4424

PHONE: (095) 556-4155

On high effectiveness of passive control by three-dimensional shock/turbulent boundary layer interactions (STBLI).

R. A. Khakimov, A. D. Khon'kin.

In the circuits of super- and hypersonic test facilities there are met elements generating STBLI and large separational zones. In our experimental investigations there are shown the high effectiveness of passive control by three-dimensional STBLI. The shock generated in front of the blunted fin interacts with boundary layer developed on the flat surface (bottom wall of the wind tunnel) on which this fin is installed. Flow control is implemented by means of air recirculation through the cavity placed beneath the interaction zone and closed by perforated (or solid in absence of any control) cover. Perforation coefficient equals 11.35% (hole diameter is 0.8 mm). No forced air evacuation from the cavity is carried out. Air recirculation flow inside the cavity is forced by pressure difference at the shock impinging on the perforated surface. The surface limiting streamlines are visualized as pictures of flowing oil drops in the interaction region near the fin. There are implemented measurements of the pressure distributions and the integral levels of pressure fluctuations. Investigation have been conducted for Mach numbers 2,3 and 4 and for unit Reynolds numbers from $9.7 \cdot 10^6$ up to $51.2 \cdot 10^6$ (changeable by variations of pressure in the plenum chamber of the wind tunnel).

The pressure distributions along the solid surface are characterized by evident "hump" associated with the interaction region. And on the case of perforated surface in front of the fin where the shock falls appropriate pressure distributions show no humps in the same places (Fig.1). These results signify that separation has alleviated and that conclusion is supported by appropriate changes in visualization pictures. Similar trends have place in relation of levels of pressure fluctuations (Fig.2).

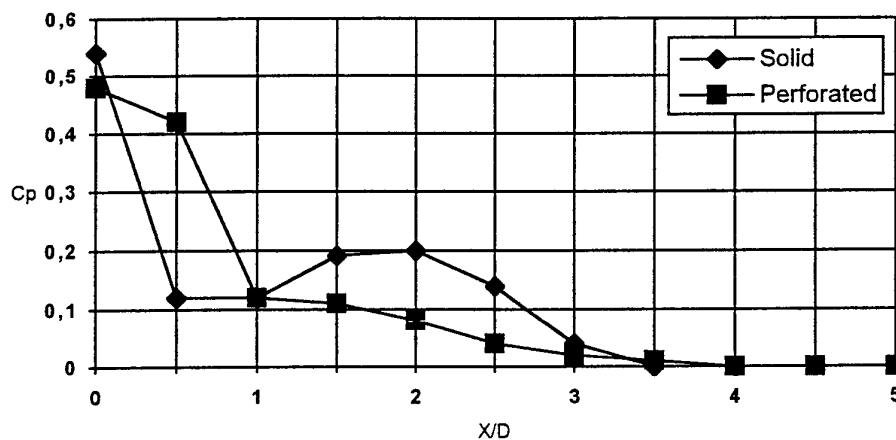


Fig.1

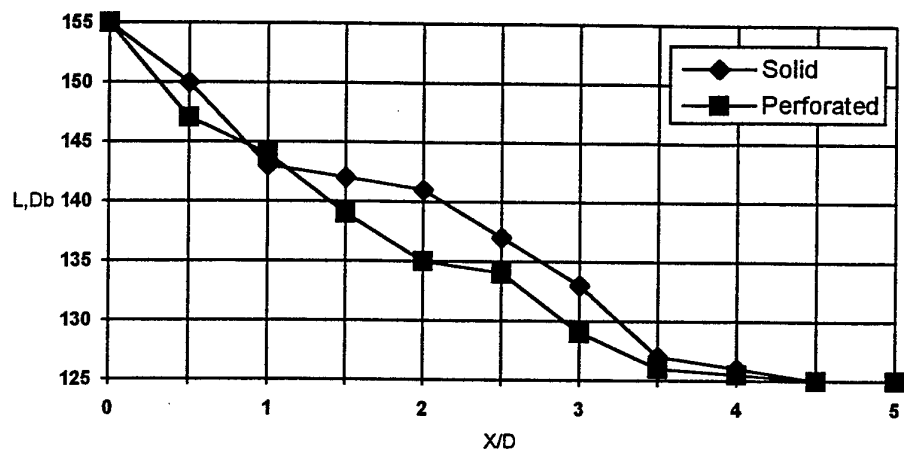


Fig.2

On marginally separated flows in dilute and dense gases

A. KLUWICK

Technical University of Vienna,
Institute of Fluid Dynamics and Heat Transfer
Vienna, Austria
e-mail: akluwick@tuwien.ac.at

Asymptotic analysis of high Reynolds number flow has shown that there exist at least two routes to the separation of a laminar boundary layer.

Firstly, a firmly attached laminar boundary-layer may be forced to separate due to the presence of a large adverse pressure gradient acting over a short distance. In contrast to classical boundary-layer theory, these pressure disturbances are induced by a local interaction process rather than imposed by the external inviscid flow. The interaction region exhibits a triple-deck structure and viscous effects are found to be of importance inside a thin layer adjacent to the wall only.

Secondly, the presence of an imposed adverse pressure gradient acting over a distance of order one on the typical boundary-layer length scale may cause the wall shear stress to decrease and finally to vanish. In general, such a solution of the classical boundary-layer equations is found to terminate in the form of a Goldstein singularity and cannot be continued beyond the point of vanishing skin friction. However, if the strength of the Goldstein singularity becomes arbitrarily small as some controlling parameter approaches a critical value, the boundary-layer thus being just marginally separated, an interaction strategy may again be applied successfully to study the separation process.

In contrast to triple-deck problems which are typically triggered by rapid changes of the boundary conditions to which the viscous wall layer is imposed, therefore, disturbances due to surface mounted obstacles, impinging pressure waves, etc. do not represent indispensable ingredients of marginal separation. Nevertheless, it is interesting and important to include such effects into the interaction theory. To this end it will be assumed that the unperturbed boundary-layer is strictly two-dimensional. In contrast, the disturbances caused by surface mounted obstacles may be two-dimensional, slowly varying in the lateral direction or fully three-dimensional. Numerical solutions of the resulting integro-differential equations obtained by means of a spectral method will be presented.

Most studies of interacting laminar boundary-layers carried out so far are concerned with flows of dilute gases where the use of the perfect gas equation of state is justified. In the past years, however, there has been a rapidly growing interest in the dynamics of fluids with high molecular complexity. These so-called BZT-fluids have the distinguishing feature that the isentropes in the pressure specific volume diagram are not strictly curved up as in the perfect gas case but exhibit regions of negative curvature in the

dense gas regime, e.g. in the general neighbourhood of the critical point. Consequently, the Machnumber no longer increases monotonically during isentropic expansion. This in turn leads to a number of new phenomena which have been studied in detail assuming inviscid flow conditions. In contrast, thermo-viscous effects have received only scant attention so far.

Analysis of these effects indicates that the existing asymptotic theories of laminar boundary-layer separation can be adapted quite easily to include dense gas flows if the disturbances of the field quantities are appropriately rescaled. Comparison with existing solutions of the full Navier Stokes equations lends support to these modified scalings. Moreover, solutions of the classical boundary-layer equations point to the existence of a new form of marginal separation caused by the non-monotoneous Machnumber variation in the external inviscid flow region.

Non-uniqueness of the separated flow past a small eccentricity elliptic cylinder rotating in viscous incompressible fluid.

G.L. Korolev

Central Aerohydrodynamic Institute(TsAGI),

140160, Moscow region, Zhukovsky-3, Russia.

E-mail:koro@buran.tsagi.rssi.ru; Phone:(7-095)179-95-78; Fax:(7-095)201-94-46

According to the asymptotic theory of viscous incompressible fluid the plane flow around elliptic cylinder that rotates at a constant angular velocity in fluid at high Reynolds number is region of interaction of the viscous layer and invicid external flow [?]. The value of a eccentricity is assumed to be order $O(Re^{-1/6})$, and flow is at rest at infinity. In elliptic coordinats fast connected with surface of body the viscous layer is described by boundary layer equations with pereodical boundary conditions for velocity profile and pressure. The pressure distribution and the circulation value around ellips are not given in advance, but they should to be determined from interaction condition and pressure pereodicity condition.

A very effective numerical method is suggested to solve the problems of interaction with peridical boundary conditions. It allows to get solution for 5-6 iterations. On the base of that method the problem of interaction of viscous sublayer and invicid external flow is solved numerically for rotating elliptic cylinder. It is shown at what parameter values connected with the Reynolds number and value of a eccentricity the flow separation takes place on the body surface. An interesting characteristic of such flow is the existence of critical parameter value. The solution does not exist if parameter value is grater than the critical value. That critical value corresponds to return point of solution and to transition on second branch of solution, which is characterized by more developed separated region. Thus it is shown, that in some parameter value change diapason it is possible the separated flows existence with different values of circulation past a rotating elliptic cylinder at one and the same parameter value.

References

- [1] V.V.Sychev, Vic.V.Sychev. Viscous flow past a rotating elliptic cylinder. *Comp.Maths Math.Phys.*, Vol.35, No.6, pp.799-804, 1995.

INFLUENCE OF A THIN INVISCID LONGITUDINAL VORTEX ON TWO-DIMENSIONAL PRESEPARATED BOUNDARY LAYER

Dr M.A. Kravtsova, Dr V.B. Zametaev

Central Aerohydrodynamic Institute (TsAGI)
1 Zhukovsky str., Zhukovsky town, Moscow region
140160 Russia

Abstract

Viscous two-dimensional boundary layer flow past a flat plate under a given unfavorable pressure gradient and influence of a thin longitudinal vortex, is considered. Let arbitrary two-dimensional inviscid flow gives an unfavorable pressure gradient along, for example, flat plate. As a result, boundary layer equations form two-dimensional marginal regime, as pressure gradient increases. The studied flow includes three-dimensional perturbation due to thin weak longitudinal vortex with given circulation Γ . Vortex line is generated by a source, which is situated at some distance upstream of flat plate leading edge, and continues along free stream line. It is supposed that height between flat plate and vortex line is about $h \ll 1$. As a result, it is possible to find solution of inviscid equation in narrow region along vortex line. This solution depends upon circulation of vortex and gives slip velocities and pressure distribution to study boundary layer. We find full three-dimensional nonlinear boundary layer problem with prescribed external velocities and pressure. But the subject of this paper is an influence of a weak vortex, which corresponds to limit $\Gamma \rightarrow 0$. Main approximation of 3D boundary layer solution in this case is two-dimensional marginal one.

Studying first approximation of 3D equation we find that a weak vortex originates during the boundary layer

development an eigenfunction of marginal type before zero skin friction point. Owing to singular behavior of the eigenfunction at marginal point weak influence will be forced and gives own contribution into interaction process and changes type of interaction. In this paper we are interesting in the case of noninteractive 2D marginal boundary layer, to study influence of interaction namely due to 3D effects. That is value of skin friction is sufficiently large and then 3D interaction starts to influence on solution.

The attachment of diffusion flames in the near wake of the fuel injector

Vadim Kurdyumov ¹, Eduardo Fernández Tarrazo ², Amable Liñán ¹

In unpremixed combustion systems the reactants are initially separated and the reactions occur simultaneously with mixing between the fuel and the air. We analyze the structure of the flow of a gaseous fuel jet mixing and reacting with a co-flowing external stream of air.

In practical applications the Reynolds number is large so that the structure of the near wake region, where the two boundary layers merge, after separation from the injector, can only be described using triple deck ideas. We shall confine our attention to the Navier-Stokes region, at the core of the lower viscous sublayer, where the effects of upstream conduction and diffusion play an important role in the stabilization of the diffusion flame. The characteristic size and velocity of this region are $l_N = \sqrt{\nu/A}$ and $U_N = \sqrt{\nu A}$, written in terms of the kinematic viscosity and the wall value A of the fuel velocity gradient.

The main parameters characterising the flow structure are the nondimensional thickness, h , of the fuel injector based on l_N , the ratio α of the wall values of the air and fuel velocity gradients and U_L/U_N , the stoichiometric flame velocity measured with U_N .

The diffusion flame is lifted off the near wake region if U_N/U_L exceeds a critical value that depends on h and α . For values of h above a critical value a recirculating region appears in the near wake that plays a significant role in the diffusion flame attachment process.

¹Universidad Politecnica de Madrid, E.T.S.I. Aeronauticos.// Plaza Cardenal Cisneros 3, Madrid, e-mail: vadim@tupi.dmt.upm.es, Tel.: 336-63-50.

²Instituto Nacional de Técnica Aeroespacial. //Torrejón de Ardoz, Madrid, e-mail: fdezte@inta.es, Tel.: 520-14-84.

Experimental Results of Unsteady Flow Characteristics in the Separated Region of Bluff Bodies

A. Leder

University Rostock, Institute of Fluidmechanics, D-18051 Rostock

Numerical procedures, e.g. Navier-Stokes calculations including $k-\epsilon$ turbulence modelling, are still failing to predict real properties of separated flows. In many cases the accuracy of drag calculations for example to design airfoils or to reduce the aerodynamic drag of automotive components are far from being applied in the technical practise. Improvements of numerical schemes can only be expected on the basis of more detailed data concerning the turbulence structures of these complex flows. Because of the non-invasive character and directional sensitivity, optical techniques like Laser-Doppler anemometry are well suited to analyse turbulent flow structures past bluff bodies.

This contribution summarizes the physical structure of separated flows and wakes behind plane and axisymmetric bodies in the incompressible flow regime. To detect the unsteady behaviours of the flowfields the velocity measurements were performed with a phase-locked Laser-Doppler system. The experimental results confirmed that the flowfields past plane and axisymmetric bodies show fundamental differences, especially in its dynamic behaviours. The amplitudes of the quasiperiodic velocity components and the Reynolds stress terms in the separated region of plane models are substantially higher than in the case of an axisymmetric flow.

The properties of separated regions like its extensions or the distributions of the turbulent terms are mainly affected by the behaviour of the free shear layers, emanating from the separation lines of the models. The characteristics of the free shear layers depend, e.g., on its geometry, the Reynolds number, the turbulence intensity, and the pressure gradient in the flow. The individual influences of these parameters on the flow structure will be discussed.

The talk ends with the discussion of an application of the presented results in the automotive industry in order to reduce the drag of passenger cars.

Leder, A.: Abgelöste Strömungen - Physikalische Grundlagen. Vieweg-Verlag, Braunschweig, Wiesbaden (1992).

Leder, A.: Dynamics of fluid mixing in separated flows; Physics of Fluids A, vol. 3, Issue 7, pp. 1741-1748, (1991).

Leder, A.; Geropp, D.: Coherent structures in axisymmetric shear Layer flows; in: Near-Wall Turbulent Flows, R.M.C. So; C.G. Speziale; B.E. Launder (Editors), Elsevier Science Publishers B.V (1993).

Modelling Complex Turbulent Flows

by

M A Leschziner, UMIST

The large majority of practically relevant flows are turbulent and characterized by complex strain associated, among others, with curvature, swirl, impingement and separation. Heat transfer, scalar transport and body forces arising from buoyancy and rotation may introduce additional complicating features. In such circumstances, turbulence modelling is a key contributor to the ability of any computational scheme based on Reynolds-averaging to provide quantitatively meaningful flow solutions.

The paper will review different approaches to computing the effects of turbulence, with attention focused mainly on the statistical framework, viewed in contrast to the simulation of turbulence. Weaknesses of simple (linear) eddy-viscosity models will be highlighted first. This will be followed by considerations of more complex types of closure which resolve the anisotropy of turbulence, either via non-linear stress-strain relations or the solution of evolution equations for all active Reynolds-stress components. Comments will be made on the physical mechanisms of interaction between various mean-flow features (curvatures, impingements, separation, rotation, heat transfer, buoyancy etc) and turbulence. Some advanced closure forms will be outlined, especially aimed at resolving complex flows which include separation, among other features.

The performance of alternative closure forms will be illustrated by reference to computational solutions for various 2D and 3D flows which are physically complex, with particular attention placed on separation. These comparisons will illustrate the sensitivity of the mean flow to turbulence modelling and convey a flavour of the challenges engineering CFD faces.

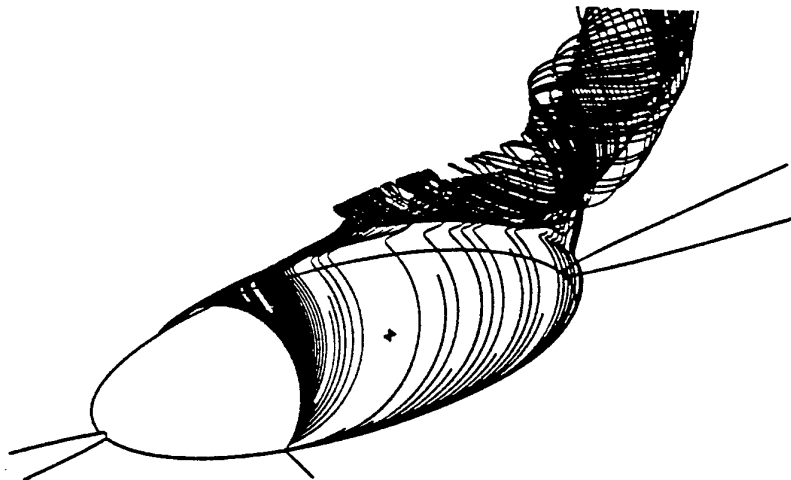


Fig. 1: Turbulent separation from a prolate spheroid at high angle of attack

Upstream influence in boundary layers 45 years ago

Sir James Lighthill
Department of Mathematics, University College London,
Gower Street, London WC1E 6BT

ABSTRACT

My two-part paper 'Boundary layers and upstream influence', published in 1953, surveyed a wide range of experimental evidence on how a disturbance in supersonic flow, which on inviscid theory would affect only downstream conditions, is able to exercise an upstream influence through the agency of a boundary layer, either laminar or turbulent. Then, by systematically comparing the data with existing attempts to account theoretically for the phenomenon, it concluded that, essentially, two mechanisms of upstream influence exist.

Mechanism (i), first suggested by Oswatitsch & Wiegardt in 1941, depended on a particular property of supersonic flow over a wall: that either wall curvature on inviscid theory, or (for a flat wall) curvature $d^2\delta_1/dx^2$ of the displacement-thickness contour on boundary-layer theory, generates a proportional pressure gradient; which, in the latter case, is $A_2 d^2\delta_1/dx^2$, A_2 being a known positive function of Mach number. Also, this positive pressure gradient might be expected to thicken the layer at a spatial rate $d\delta_1/dx = A_1 (A_2 d^2\delta_1/dx^2)$ where A_1 , although far from precisely known, must be less for a turbulent than for a laminar layer; so that, finally, the e-folding distance of upstream influence would be $A_1 A_2$.

Mechanism (i) was compared in Part II of my paper with a different proposal (Howarth 1948) for a theoretical programme concerned with 'propagation up the subsonic layer', in which only the undisturbed boundary-layer distribution (including its subsonic part) would be taken as influenced by viscosity while disturbances to it would be treated inviscidly. The reason why attempts to carry out this programme had failed was explained in terms of earlier theories of boundary-layer instability, in which time-dependent disturbances had been found to be influenced by viscosity in two layers: a wall layer and a critical layer. For disturbances independent of time these would coincide into a single wall layer where, however, the influence of viscosity still needed to be taken into account; in which case, the analysis could be satisfactorily completed but became in essence merely an expression of mechanism (i) with a relatively precise determination of A_1 .

Mechanism (ii), identified in work by Lees (1949) at Princeton and by Liepmann, Roshko & Dhawan (1949) at Caltech, depended on the upstream spreading of a separation bubble till it became sufficiently slender to cause no further separation ahead of it. Part I of my paper was concerned to point out that, although mechanism (i) can work only when a well-defined coefficient A_2 exists (that is, for supersonic flow), mechanism (ii) is effective in both subsonic and supersonic flow. This was illustrated by analysing data on flow up a step at various Mach numbers (with various locations for transition to turbulence) in terms of boundary-layer separation studies. Those instructive examples, which may today be somewhat less known, and which included several interesting cases of both steady and also unsteady separated flows, can appropriately be recalled in a colloquium devoted to such phenomena.

The Local Laminar Separated Flows Near Surface Distortions

Igor I. Lipatov, Igor V. Vinogradov
Central Aerohydrodynamic Institute (TsAGI)
Zhukovsky-3, Moscow Region, 140160, Russia
Phone: +7-095-5564372, Fax: +7-095-5564337
E-mail: igor@lipatov.rosmail.com

The surface distortions are used to delay laminar-turbulent transition . Experimental investigations allowed to obtain some integral characteristics of distortions influence on the transition but the local flow structure and basic physical mechanisms leading to separation and other important phenomena are not yet completely understood.

The main problem is connected with basic physical mechanisms acting in separated flows. Real flows are characterised by high Reynolds numbers and this justify application asymptotic methods of analysis.

There are known four regimes of local separated flows near surface distortions: 1- laminar; 2- unsteady laminar (coherent in time and/or in space); 3- transitional; 4- turbulent. Analytical and numerical analysis of the local flows near distortions placed in the laminar boundary layer were published in many papers (see for example surveys done by Smith, Duck, Bogolepov etc.). It was shown the mathematical models for such flows strongly depend on the correlation between geometrical sizes of distortions and the boundary layer thickness. Asymptotical analysis have led to the general scheme of locally disturbed shear flows.

The local flow near distortion placed in the laminar boundary layer may be described in general case by three-dimensional unsteady: i) incompressible Stokes equations ,ii) incompressible Navier-Stokes equations, iii) interactive boundary layer equations with some form of induced pressure gradient, ii) original boundary layer equations. In this report analytical and numerical results are presented for some local flow regimes near three-dimensional distortions corresponding to the case (iii). These results describe the velocity profiles along with the surface skin friction and pressure distributions. Influence of the wall temperature on the flow structure was investigated.

New results are obtained for unsteady local separated flows. This regime is the least investigated now. It may be supposed some surface exists in space of geometrical parameters corresponding to the boundary between steady and unsteady local flows. The flow type change (steady -unsteady) is connected with the reversed flow non-linear stability characteristics. Numerical results for two-dimensional flows obtained by Kazakov, Ryzhov et al and in this report show the stationary flow for a hump with a fixed geometry exists only if hump's height don't exceed some critical value. For larger height values selfoscillations arise with periodical (quasi periodical) reversed zones formation.

Euromech 384

Colloquium on Steady and Unsteady Separated Flows

University of Manchester, 6-9th July, 1998.

**Wake formation around islands in oscillatory laminar
shallow-water flows**

P.M.Lloyd, P.K.Stansby and D.Chen.

Hydrodynamics Research Group, School of Engineering, University of
Manchester.

The stability of a wake generated by current flow past a circular cylinder in shallow water is now well understood and may be defined by a stability parameter ($C_f D/h$, where C_f is the friction coefficient, D is diameter and h is water depth). In this study we investigate experimentally the wake generated by oscillatory shallow-water flow around a cylinder in the simple situation of laminar flow. The flow is defined by an 'amplitude' parameter or Keulegan-Carpenter number ($U_0 T/D$ where U_0 is velocity amplitude and T is oscillation period). The wake may be classified as symmetrical without pairing, symmetrical with pairing, sinuous or vortex-shedding and stability is not a well-defined phenomenon. The wakes are visualised by dye and particle-tracking velocimetry. Related results for a conical island of 8° slope are also presented with computational comparisons.

FLOW LIMITATION AND SELF-EXCITED OSCILLATIONS IN COLLAPSIBLE CHANNELS

X.Y.Luo

Dept. of Engineering, Queen Mary & Westfield College,
University of London, U.K.

The study of flow-induced oscillation in tubes and channels has important applications in physiology, notably in the context of expiratory wheezing in airways. One of the principal questions of interest in the forced expiration from the lungs is whether there is a causal relationship between wheezing (a kind of oscillation) and "flow limitation", i.e. the phenomenon that increasing a large expiratory driving pressure does not lead to an increase in flow rate on account of airway collapse. It is an attempt to answer this question that stimulates the current study.

The flows and oscillations are simulated numerically in a finite-length collapsible channel where part of the wall is replaced by a membrane, with or without wall inertia. The Navier-Stokes equations and membrane equation are solved simultaneously using the finite element method. Both steady and unsteady flows are studied, with control parameters such as the upstream and downstream transmural pressures, longitudinal membrane tension, and wall inertia parameter. By increasing the pressure drop inside the channel, and calculating the flowrate as an output, we investigate at which parameter values flow limitation occurs and, if it does, whether self-excited oscillations would follow. Compared with our previous study where the downstream transmural pressure and upstream flowrate are fixed, it is found that the current system is numerically more unstable and a new numerical approach has to be applied to investigate the instabilities of this system. Preliminary results are presented to explore the relation between the flow limitation and self-excited oscillations, and the main conclusion seems to be that the occurrence of flow limitation is not a sufficient condition for self-excited oscillations if a membrane with small inertia is used as the wall model.

Abstract for EUROMECH 384

Vortex Dynamics in the Wake of a Sphere

Rajat Mittal†

Department of Mechanical Engineering

237 MEB, University of Florida

Gainesville, FL 32611 USA

Ph (352) 392-6751, (F) (352) 392-1071

mittal@gollum.me.ufl.edu

The study of axisymmetric bluff-body wake flows has relevance in aerodynamics, marine hydrodynamics and particulate flows. The wake of a sphere which can be considered a prototypical axisymmetric bluff-body is nowhere as well understood as its two-dimensional counterpart, the cylinder. A DNS study has therefore been undertaken to gain better understanding of the vortex dynamics in the wake of a sphere at low to intermediate Reynolds numbers. The solver that has been developed employs an accurate Fourier-Chebyshev collocation method in conjunction with a fractional step scheme to solve the 3-D incompressible, viscous, unsteady flow past a prolate spheroid. Flow past a sphere can be solved as a special case. The series of simulations carried out show that as the Reynolds number is increased, the sphere wake undergoes a sequence of bifurcations and these confirm some of the observations made in previous experimental studies. In addition, the current simulations afford us an in-depth look at the topology and formation mechanism of the vortex structures and a number of unique features including symmetric vortex shedding and formation of vortex rings are described in detail. Furthermore, by comparing our observations of sphere and cylinder wakes we are able to point out some interesting similarities and differences between the two flows. Finally, results of our recent study on the response of the sphere wake to free-stream fluctuation will also be presented. This study has been motivated by its relevance in particle-turbulence interaction and fluid-structure interaction in marine applications. Results to be presented will focus on describing the effect that transverse free-stream velocity fluctuations have on vortex shedding and the hydrodynamic forces experienced by the sphere.

†Assistant Professor

On the nonlinear growth of two-dimensional Tollmien-Schlichting waves in a flat-plate boundary layer.

J. Moston, P.A. Stewart & S.J. Cowley

Abstract

This paper re-examines the nonlinear development of two-dimensional Tollmien-Schlichting waves in an incompressible flat-plate boundary layer at asymptotically large values of the Reynolds number. Following previous authors, we restrict attention to the 'far-downstream lower-branch' régime where a multiple-scales analysis is possible. Then to leading-order the waves are inviscid and neutral, and governed by the Davis-Acrivos-Benjamin-Ono equation (Zhuk & Ryzhov 1982, Smith & Burggraf 1985). The DABO equation has a well-known three-parameter family of periodic solutions, the large-amplitude (soliton) limit of which bears a qualitative resemblance to the 'spikes' observed in certain 'K-type' transition experiments (Kachanov, Ryzhov & Smith 1993). The variation of the parameters over slow length- and time-scales is controlled by a viscous sublayer. For the case of a purely temporal evolution, it is shown that a solution for this sublayer ceases to exist when the amplitude reaches a certain finite value as a result of a quasi-steady separation. For a purely spatial evolution, it appears that an initially linear disturbance does not evolve to a fully nonlinear stage of the envisaged form because of steady separation. The implications of these results for the 'soliton' theory of spike formation are discussed.

TURBULENT SHEAR LAYERS AND STABILITY THEORY

by

P. K. Sen and Srinivas V. Veeravalli

Department of Applied Mechanics

Indian Institute of Technology, New Delhi - 110016; INDIA

This paper investigates the role of hydrodynamic stability theory in wall-bounded turbulent shear flows, especially as this theory has a significant role in free turbulent shear flows. See for example the comprehensive review by Liu [1] and the more recent overview by Roshko [2]. Pioneering ideas were given by Malkus [3]. However, significant advance was made in using hydrodynamic stability theory in analysing wall bounded turbulent flows in the works of Reynolds & Tiederman [4], followed by Reynolds & Hussain [5]. Reynolds & Hussain [5] derived an extended Orr-Sommerfeld equation which captures the evolution of an organised disturbance in wall-turbulent flow, based on the Newtonian eddy model. However, although this extended Orr-Sommerfeld equation yields near neutral modes, it does not exhibit instability. Here we present a further extension to the Orr-Sommerfeld equation using a non-isotropic eddy viscosity model along the lines of Pope [6], which does support unstable modes and mimics some of the key features of wall-turbulence. The extended Orr-Sommerfeld equation obtained is as follows:

$$\begin{aligned} i\alpha[(\bar{u} - c)(\phi'' - \alpha^2\phi) - \bar{u}'\phi] - 1/R[\phi'''' - 2\alpha^2\phi'' + \alpha^4\phi] \\ - 1/R[E\{\phi'''' - 2\alpha^2\phi'' + \alpha^4\phi\} + 2E'\{\phi''' - \alpha^2\phi'\} \\ + E''\{\phi'' + \alpha^2\phi\}] + \frac{\lambda E}{R}[-2i\alpha\phi''' + 2i\alpha^3\phi'] \\ + \frac{2i\alpha\phi'}{R}[\lambda E'' + 2\lambda'E' + \lambda''E] = 0. \end{aligned} \quad (1)$$

In above, E is the eddy viscosity, and λ is the anisotropy function obtained from Pope's model. (Details of derivation may be seen in [7]). The above equation yields unstable wall modes, characteristic features of which scale with the inner variables of turbulent flow.

The experimental results for the measured compensated one-dimensional power spectra of the u' velocity fluctuations, viz. $\kappa^+ E_{11}^+(\kappa^+)$, were plotted against κ^+ . Here $\kappa \equiv 2\pi f/\bar{u}$, where f is the frequency. The spectra were obtained at different y^+ in the buffer and log regions. These were compared with the growth-rate curve $\alpha^+ c_i^+$ versus κ^+ obtained from eq. (1). The entire growth-rate curve, i.e. the entire range of unstable α^+ , is contained within the experimental power spectra. Other key features, like peaks of r.m.s. velocity, shear stress and production, are also mimicked by the instability waves.

At present the ideas are being extended to flows with adverse pressure gradient, including separated flows. The work is supported by C.S.I.R. (India) grant nos. 22 (229)/ 92 EMR-II and 22 (254)/ 96 EMR-II

References

- [1] Liu, J.T.C., 1988, Adv. App. Mech., **26**, pp 183-309.
- [2] Roshko, A., 1992, 'Theoretical and Appl. Mech.', ed. Bodner *et al*, publ. Elsevier Science.
- [3] Malkus, W.V.R., 1956, J. Fluid Mech., **1**, pp 521.
- [4] Reynolds, W.C. and Tiederman, W.G., 1967, J. Fluid Mech, **27**, part 2, pp 253-272.
- [5] Reynolds, W.C. and Hussain, A.K.M.F., 1972, J. Fluid Mech, **54**, part 2, pp 263-288.
- [6] Pope, S.B., 1975, J. Fluid Mech., **72**, part 2, pp 331-340.
- [7] Sen, P.K. and Veeravalli, S.V., 1998, to appear in *SADHANA*.

The Some Aspects of Three-Dimensional Separated Flows

Yu. D. Shevelev

Institute for Computer Aided Design of Russian Academy of Sciences
19/18, 2-nd Brestskaya, ICAD, Moscow, Russia, 123056
e-mail: shevelev@inapro.msk.su; tel: (095) 2509280 ; fax: (095) 2509554

1. Abstract

This paper summarises the recent progress of author and his colleges with methods for modelling and prediction of 3-D separated flow. It also describes some of the last research using dynamically system theory being carried out. The topological structure of separated flows considered. The different physical models of separation investigated. A balanced approach involves both theoretical and numerical studies. Finally it displays some of the applications of the methods and comparison of results of experimental and theoretical research.

2. Introduction

Basically our understanding of nature of 3-D separated flow based on analysis of flow visualisation by different methods. In spite of great progress in computational fluid mechanics the main understanding of viscous separated fluid flow near wall and in jets are far away from final solution. The concept of "separation" and "separated flows" devoted the proceeding of the symposiums ([1]), special reviews ([2]) and others.

The separation occurs in subsonic, supersonic flows for both laminar and turbulent regimes. The real picture of 3-D separation on blunt bodies, real aircraft configuration depend on geometry, angle of attack, Mach and Reynolds number, temperature and another factors. The structure of separation depend on balance of viscous stresses, gradients of pressure, convection, secondary flows and unsteadiness. The influence of different forces and concurrent factors produces the large eddy structures. These structures are conditionally stable and self-governing ones. The 3-D separated flows have the same similarities for different parameters of flow and different cases.

In previous years we developed an analytical method for the computation of incompressible and compressible 3-D boundary layer problems based on the method of successive approximations. We obtain the analytical solutions in first approximation as formulas ([3-4]). Using the analytical solutions it's possible to estimate the position of "separation line" and character of singularity near "points of separation". We developed the finite-difference methods of solution of laminar and turbulent boundary layer theory problem ([4,5,6]). The numerical calculation of steady boundary layer problem by marching method permit to receive solution until "separation". The strong and weak interaction of external flow and boundary layer produced the pressure redistribution and fluid flow depend on induced pressure.

3. Mathematical models

Let us consider the singularities arising in viscous three-dimensional steady fluid flow past arbitrary smooth geometry S . The streamlines are particle path lines. The characteristic determinant of full Navier-Stokes equation generated along the

streamlines surface. In the framework of 2-D boundary layer the separation means that the wall friction is equal zero and the limiting streamlines leave the body surface. The singular points we define as points where the all components of velocity (or skin friction components on wall) are equal to zero. We consider two cases of singular points: 1) near surface; 2) in free flow field. The limiting streamlines on the surface and the vector field of skin friction on the surface are the same ones.

Suppose that the components of velocity vector U can be expanded in a Taylor series. By substitution of the expansion in Navier-Stokes equation we obtain the relationships between unknown coefficients. The streamline equations employing these velocity components are first order differential equations and can be solved using a marching procedure. The initial conditions determine a streamlines that begin in a prescribed downstream points. Another possibility can be realised if 3-D solution of the Navier-Stokes equation can be obtained by numerical method. Thus the velocity components are known. And the pattern of streamlines can be displayed.

The system of streamline equation by transformation can be reduced to canonic form. We took into account the influence of non-linear terms which can appear in the expansions of component of velocity near wall. The common mechanism of separation studied. The combinations nodes, saddles, focuses points produced the different pattern flow near singular points. The singularity of focus type are formed by flow from different parts of boundary layer and after separating the particles propagate as concentrated vortex line or jet. Classification of singular points for linear and nonlinear case obtained. Different generic forms of bifurcation considered and the influence of terms of high order took into account. Topography of streamlines in 2-D section of 3-D flows is classifying. Questions of structurally stable systems and different possibility of stability losses discussed.

4. The numerical methods

a) The solver for predicting of unsteady three-dimensional incompressible fluid flow developed. Our method based on using of arbitrary nonorthogonal co-ordinate system for governing differential equations. By utilising of tensor analysis it's possible to obtain of unsteady 3-D Navier-Stokes equation in invariant form independently on the choice of co-ordinate system. When the tensor differential equations used then as a result the invariant form of finite -difference or finite-volume equations will be obtained . It's means that one computer program now will be available for different problems.

The numerical solution of full Navier-Stokes equation set based on combination of the operator-splitting technique or the fractional step schemes. The time step is divided on three subsets. First step is solution of convection part of equations subject to corresponding initial and boundary condition. Two version of codes realised for calculation of pressure on second step: one based on pseudo compressible approach and another one on solution of Poisson equation for pressure. The third step take into account the influence of viscous terms. As test for codes used the analytical solution which obtained by mathematical models.

b) For numerical investigation of separated compressible viscous gas flow past the sharp bodies at the angle of attack the finite volume method developed. An adaptive generation grid technique used to have the ability to choose the distribution of points near surface boundary and clustering of points at regions where strong variations of the flow and the curvature geometry occur. Stars with different cross section studied by using conform mapping to cone. Separated flow past elliptical cones, wings, stars for different regime flow considered. A good correspondence with experimental data

(position of separation, reverse-circulation zones, pressure and heat distribution) and boundary layer theory obtained.

5. Results

a) Analytical studies of non-linear interaction of singularities focuses on understanding and controlling separation and vortex development in 3-D flow. Main features of flows in separated zones are classifying and some examples of its application will be given. Simple analytical models with 6 singularities compared with flow separation pattern observing experimentally on surface on the downwind side of a blunt body at an angle of attack. The same results obtained by numerical integration of Navier-Stokes equation.

b) The 3-D three-dimensional viscous supersonic flows past sharp bodies on windward part are in good agreement with experimental data and boundary layer theory. Structure of flow on leeward part in separation region characterised by boundary layer-shock wave interaction and we can show the results of comparison experimental data and results of numerical calculations.

Our results based on experimental studies of the 3-D surface flow patterns. Experimentally it was analysed and generalised the 3-D flow pattern flow past segmental conic bodies (type of "Soyez", "Appolo"), wings for Mach number 3-6 and angles of attack 3-90 degree ([7]).

Bibliography

1. Separated Flows and Jets, IUTAM-Symposium, 1990, Springer-Verlag, Novosibirsk
2. A.E.Perry, M.S.Chong A Description of Eddying Motion and Flow Patterns Using Critical-Point Concepts, Annual Review of Fluid Mechanics, 1987, 19, 125-155
3. Shevelev Yu. D. Three-Dimensional Problems of Laminar Boundary Layer, monograph, Moscow, "Nauka", 1977, 223 p. (in Russian)
4. Shevelev Yu. D. The three-dimensional problems of computational fluid dynamics, monograph, Moscow, "Nauka", 1986, 367 pp. (in Russian)
5. Shevelev Yu. D., Aleksin V. A. The Three-Dimensional Turbulent Boundary Layer on a Body of Complex Shape Izv. Akad. Nauk SSSR, Mekh. Zhidk. Gaza, No 5, 1986, pp.25-35
6. Shevelev Yu. D., Kravchenko V. I., Schennikov V. V. Numerical investigation of Unsteady Viscous Incompressible Flow Lecture Notes in Mathematics, No 771, 1979, pp. 323-359, Springer-Verlag
7. Shevelev Yu.D., Glagolev A.I., Dudin G.N., Zubkov A.I., Svirin B.N. Mathematical Modeling of 3-D Separated Flows, Proceedings of 7 All-Russian seminar "Modern Problems of Mathematical Modeling", 1997, pp.160-163

Repercussions from Goldstein's 1948 paper

Some of the modern repercussions from Sydney Goldstein's landmark 1948 paper will be described, after a brief summary of the steps in-between (on the numerical verification, Stewartson's non-removal theory, cases of removal, marginal separation; and the alternative strategy of large-scale separated flow with pressure-displacement interaction).

The main repercussions of interest here are in multi-blade-wake motions and their interactions for two- and three-dimensional flows within rotary boundary layers, in motion over two- and three-dimensional roughnesses buried inside a viscous layer, in unsteady boundary layer evolution, and in large-scale separations. First, in blade-wake flows, with nearly aligned configurations of thin blades (airfoils), there are several novel features.

1. Multiple interactions between each viscous blade- and wake-flow.
2. Inner-outer viscous-inviscid interaction at remarkably small incidence.
3. Double viscous layer interactions.
4. Pressure-displacement interactions covering entire blades.
5. Long-scale separations.
6. Unsteadiness, and transitions in the wakes.

The applications are to helicopters, rotorcraft, fans, turbine flows, domestic and garden appliances, nature, industrial mixers. Second, in flows over surface-mounted roughnesses, the items of concern are these.

1. Numerical solutions in two and three dimensions.
2. Flows induced by steep edges.
3. Upstream separation (e.g. for a forward facing step, Gurney flap) and downstream separation (e.g. backward facing step).
4. Vortex motion near the wing-tip of a roughness.
5. Origin of horseshoe vortices.
6. Unsteady flows, and transitions.

There are also many developments in unsteady boundary layers, such as secondary instability and interactions, and in large-scale separations, such as bluff-body motion and flow over a steep roughness.

High Reynolds number perturbed flows with a vanishing skin friction

Victor V. Sychev, Jr.

Central Aerohydrodynamic Institute, Moscow

Steady flow of incompressible fluid over a small 2-D humps placed on a straight walls of a divergent channel is considered. For the undisturbed flow we take the exact Jeffery-Hamel solution with the divergence angle close to its critical value, when the skin friction (both positive and negative) becomes small. The triple-deck structure appears near the humps: locally inviscid main part of the flow and thin viscous sublayers on the walls. Different regimes depending on the skin friction value and humps dimensions are considered. The solutions of the linear problems give an interesting streamlines patterns for separated flows in wall layers. The same problem was considered for unsteady case, when the form of a humps depends on time. It was shown that the solution of the initial value problem tends at large time to its periodic or steady state only for positive values of the skin friction. For negative values of skin friction it appears an exponential growth and the solution does not tends to periodic or steady state, in spite of such a solutions exist.

Outer steady flow over 2-D hump, when undisturbed boundary layer with a vanishing skin friction is described by appropriate self-similar Falkner-Skan solution, is investigated. In detail it was considered the triple-deck structure with a given pressure gradient, determined by the shape of the hump. As the result of the solution of the linear problem it was determined the existence of critical value of the height (depth) of the hump $h = h^*$ when the skin friction become zero for the first time at some point S. For h greater than h^* it appears Goldstein singularity. When $h = h^*$ there exist two solutions. One of them with the removable Ruban singularity [1] in S and other - smoothly passed through this point. The last is in a full agreement with Korolev numerical results [2] for some boundary layer problems. At the same time our results contradict to the concept of the paper [3], since according to [3] the skin friction tends to zero as a square of a distance from point S, whereas according to [1] it tends to zero as a module of this distance.

References

1. Ruban, A.I. 1981 Izv. AN SSSR, Mech. Zhid. Gaza, 6, 42-52.
2. Korolev, G.L. 1991 Zh. Vychisl. Mat. Mat. Fiz., 31, 1706-1715.
3. Bogdanova-Ryzhova, E.V. and Ryzhov, O.S. 1994 Theor. Comput. Fluid Dyn., 6, 193-212.

Separation in two-fluid flows.

S.N. Timoshin
Department of Mathematics
University College London
Gower Street
London WC1E 6BT

Email: sergei@math.ucl.ac.uk
Tel: 0171-504-2205
Fax: 0171-383-5519

A high Reynolds number asymptotic approach is used to model flow separation and related phenomena in two-fluid laminar flows. Two examples of the flow geometry are considered: a flat plate boundary layer on a thin film of a different fluid, and a two-fluid Poiseuille flow in an inclined channel. A multi-zone flow scheme is developed for such flows, with the effects of a fluid-fluid interface incorporated into a near-wall viscous flow zone. Numerical solutions for the flow near a localized wall roughness are presented for the cases of on-wall and off-wall separation, and for both supercritical and subcritical (in terms of upstream propagation of disturbances) flow regimes. A theory of transcritical flows involving two fluids is developed in some detail.

The Absolute Instability Of Thin Wakes In A Compressible Fluid.

M Türkyılmazoğlu, J S B Gajjar & A I Ruban

April 22, 1998

*Mathematics Department, University of Manchester, Oxford Road,
MANCHESTER M13 9PL, U.K.*

Abstract

The absolute/convective instability of two-dimensional wakes forming behind a flat plate and near the trailing-edge of a thin wedge-shaped aerofoil in a compressible fluid is investigated. The mean-velocity profiles are obtained by solving numerically the classical compressible boundary layer equations with a negative pressure gradient for the flat plate case, and the triple-deck equations for a thin wedge-shaped trailing-edge. In addition for a Joukowski aerofoil the mean boundary layer flow ahead of the trailing-edge was also calculated. A linear stability analysis of the boundary layer profiles shows that a pocket of absolute instability occurs upstream of the trailing-edge with the extent of the instability region increasing with thicker aerofoils. The region of absolute instability persists along the near-wake axis, while the majority of the wake is convectively unstable. For a thin wedge-shaped trailing-edge in an incompressible fluid, a similar stability analysis of the velocity profiles obtained via a composite expansion, also shows the occurrence of absolute instability ahead of the trailing-edge for a wedge-angle greater than a critical value. For increasing values of the wedge-angle separation also occurs near the trailing-edge and the extent of absolute instability increases.

Calculations also show that for an insulated plate compressibility has a stabilizing effect in terms of absolute instability. Moreover cooling the wall destabilizes the flow unlike wall heating.

SEPARATED AIRFLOW OVER LAUNCH VEHICLE NOSE PART

A. Vaganov, V. Plyashechnik, D. Ivanov, I. Yegorov and S. Zadonsky
Central Aerohydrodynamics Institute (TsAGI),
Zhukovsky, Moscow Region, 140160 Russia

Experimental and numerical study of an airflow over launch vehicle nose part is considered. Experimental study were done with the different wind tunnels of the Central Aerohydrodynamics Institute (TsAGI) and different scale models. The description of wind tunnels parameters and devices used is given in this paper. The compressible Reynolds-averaged Navier-Stokes equations are used for numerical solution of this problem.

The calculations were made for two wind tunnels flow conditions and two scale models ($M_\infty=0.6-3$, $Re_{\infty,L}=10^6-10^7$) as well as for flight conditions and flight scale vehicle ($M_\infty=0.6-3$, $Re_{\infty,L}=10^7-10^8$). To analyze the appearance and development of the separated zones on a body surface the plots of skin-friction coefficient along body surface are presented on the Fig. 1. These data illustrate the presence of the separated zone in the base area of a nose part of the body for different Mach and Reynolds number under considerations.

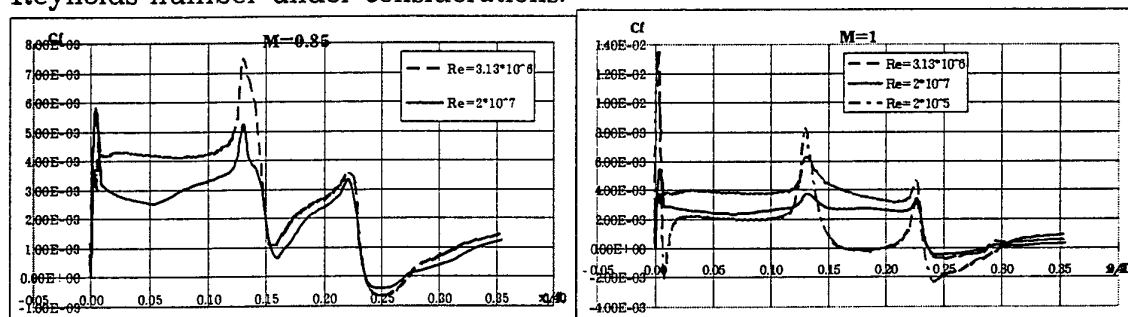


Fig. 1 Skin-friction coefficient along body surface (numerical data)

There are three closed separation zones for small Reynolds numbers. For validation of these numerical data the experimental results are shown on the Fig. 2. The light zones are corresponded to separated zones.

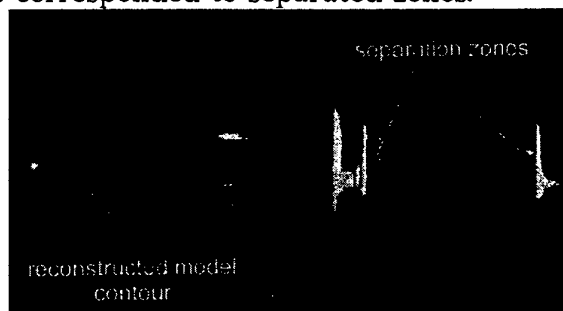


Fig. 2 Experimental data ($M_\infty=1$, $Re_{\infty,L}=10^6$).

Experimental data of a local pressure distribution, photo of flowfields are compared with numerical results. The problem of wind tunnels data recalculation on the flight conditions ones is analyzed.

The experimental data of this paper were obtained on the base of Central Aerohydrodynamics Institute and Lockheed Martin Aeronautics Company combine researches.

3-D BOUNDARY-LAYER FLOW PAST A CUSP SECURED TO A FLAT PLATE

Gregory G. Vilensky

Research and Production Enterprise MARINE EQUIPMENT
46, Primorsky Prospect, St. Petersburg, Russia 197374.
Email: vilensky@mail.admiral.ru Tel/Fax: +7 (812) 2680529

The work deals with the incompressible 3-D boundary-layer flow in the streamlined corner formed by a semi-infinite flat plate and a cusp secured vertically to the plate's surface so as to ensure that the leading edges of the cusp and the plate intersect at a right angle. The analysis focuses about the structure of the plate's boundary-layer. If the thickness of the cusp were zero, the boundary-layer flow developing at the plate would be 2-D. However, the presence of the cusp sets up a singular pressure gradient which induces a cross-stream velocity perturbation in the plate's boundary layer. The purpose of the present study is to examine flow adjustment in the vicinity of the curvilinear rib of the corner formed by the wall of the cusp and the plate.

The major finding of the work is the description of the asymptotic structure of the plate's viscous flow. It is shown to consist of three characteristic domains: an outer region, an intermediate region and an inner region in the immediate vicinity of the cusp. The outer region occupies most of the plate's surface, save a thin strip near the cusp. The flow is quasi-two-dimensional here, with the cross-stream momentum equation being linear. Since the component of the velocity orthogonal to the cusp's wall does not vanish in it, there arises an intermediate region. It has the same thickness as the outer region. However, the growth of the cross-stream perturbations near the cusp makes the governing equations fully non-linear. The analysis of these equations reveals the existence of the inner non-linear region near the cusp's wall, in which the velocity component orthogonal to the cusp must decrease to zero. The solution is a power series in normal distance to the cusp's wall here. The boundary-layer thickness in the inner region is much smaller than in the above ones. In summery it is suggested that presumably the viscous flow over the above plate consists of two different boundary layers. One of them originates from the plate's leading edge and evolves through the outer region towards the wall of the cusp. The other one is driven by the pressure gradient created by the curvature of the cusp's wall and develops in the immediate vicinity of the cusp towards the outer region. The two flows overlap in the intermediate region.

EUROMECH 384: Colloquium on Steady and
Unsteady Separated Flows

Flow separation behind steep-sided obstacles

Andrew G. Walton

Mathematics Department

Imperial College of Science, Technology & Medicine

180 Queen's Gate, London SW7 2BZ, U.K.

Abstract.

The landing of a helicopter on the flight deck of a ship is a complicated task which is rendered more difficult by air wake effects. The air flow pattern over the flight deck is hard to predict due to the complicated surface geometry and the motion of the ship.

The aim of this work is to consider a simplified model and by using a combination of high Reynolds number asymptotics and numerical computation, calculate the flow field in the vicinity of the landing pad. The model proposed here is a condensed form of triple-deck interaction and can accommodate separation and reattachment of the boundary layer.

In its simplest form, the problem is that of a two-dimensional boundary-layer flow over a backward-facing step. Extensions to the theory provide for more complicated geometries and allow us to study how the location of the separation and reattachment points varies with changes in the surface geometry.

The work is closely related to that of Smith & Walton (Proc. Roy. Soc. Lond. A454, pp. 31-69, 1998) concerning flow over steep-edged obstacles and is being carried out in association with DERA Bedford.

Tel: 0171 594 8514 Fax: 0171 594 8517 email: a.walton@ic.ac.uk

SEPARATED FLOW IN THE SUPERSONIC INLET

I. V. Yegorov, D. V. Ivanov, V. P. Starukhin
Central Aerohydrodynamics Institute (TsAGI),
Zhukovsky, Moscow Region, 140160 Russia

E-mail: cfd@tsagirssi.ru, tel.: (095) 556-6188, fax: (095) 556-4337

Flow in a supersonic inlet is one of the different applications where we deal with separated flows. Boundary layer separation changes sufficiently the main features of an inlet flow. Numerical simulation of these flows is very difficult. By the first, it is connected with laminar-turbulence transition and separation of the flow. Particularly, at near calculation inlet conditions when shock wave from central body puts on forward edge of outer part of inlet a separation of boundary layer with laminar character of a flow and follow turbulence attachment are realized. These phenomena are often take place in aerodynamics because a laminar-turbulence transition is "earlier" on a free surface appearing through a separation of boundary layer.

Investigation of separated flow in simplest two-dimensional supersonic inlet is considered in this paper. Numerical simulation [1] is fulfilled on the base of Reynolds averaged Navier-Stokes equations with the using of two equations $q-\omega$ turbulence model [2]. The choice of base calculation conditions is connected with experimental data obtained in TsAGI. For example, the comparison between numerical and experimental data are presented in the Figs. 1, 2. Mach and Reynolds numbers are determined from inlet entrance parameters, $M_\infty=3.4$ $Re_L=2 \cdot 10^6$.

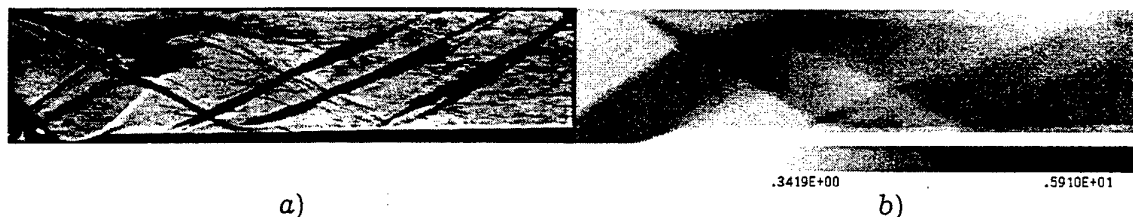


Fig. 1. Comparison of experimental (a) and calculation data (b). a) - Tepler camera picture, b) - density (ρ/ρ_∞) field.

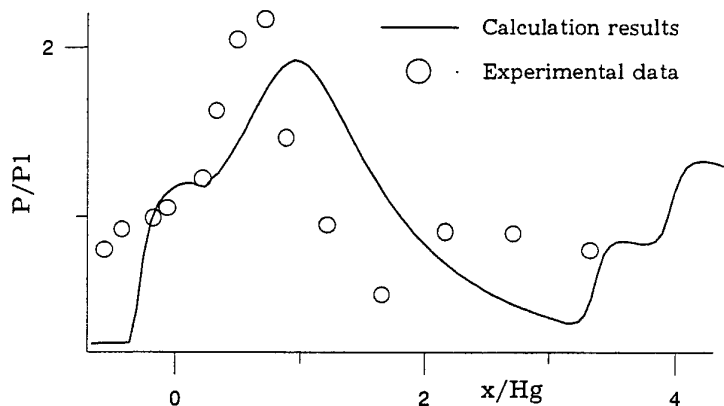


Fig. 2. Normalized pressure distribution along upper solid wall.

References

1. Ivanov D.V., Obabko A.V., Yegorov I.V. "Simulation of separated flows on the base of differential turbulence model". AIAA Paper, 97-1861. 1997, pp. 1-11.
2. Menter F.R., Rumsey L.C. "Two-equation turbulence models for transonic flows", AIAA Paper 94-2343. 1994.

BOUNDARY LAYER FREE-INTERACTION THEORY FOR TRANSONIC FLOW

Vladimir I. Zhuk

Computer Center of the Academy of Sciences, 117967 Vavilova, 40, Moscow, Russia
e-mail: zhuk@ccas.ru ; telephone: (095)1356239 ; fax: (095)1356159

The generalization of a triple-deck theory to the case of transonic velocities can lead to differing estimates of the scales of the perturbations depending on the role of unsteady effects. The problem with interaction in the transonic range, considered for the first time in [1], admits of the introduction of time in the external potential flow domain (upper deck) [2] without changing the estimates corresponding the stationary conditions. Another mechanism for the wave propagation has been proposed in [3], where the scales of the variables introduced necessitate the retention on the time-dependent terms in the upper deck as well as in the lower deck.

Below, we describe an asymptotic analysis of non-linear perturbation, the amplitude of which exceeds the value assumed in [3]. The concept of self-induced pressure in the case of such amplitudes prescribes the scales of variables which is different from that in [1]—[3], with the flow acquiring a four-deck structure. The study of the two-dimensional velocity field reduces to solving an integrodifferential equation for a function which depends on time and one spatial coordinate. The theory which is developed achieves a continuous transition from subsonic to supersonic flow since the above-mentioned governing equation contains the Burgers equation and the Benjamin-Ono equation as limiting cases.

References

1. *Messiter A. F., Feo A., Melnik R. E.* Shock-wave strength for separation of a laminar boundary layer at transonic speeds // *AIAA Journal*. 1971. V. 9. N. 6. P. 1197–1198.
2. *Ryzhov O. S.* On the unsteady boundary layer with a self-induced pressure at transonic external flow velocities // *Dokl. Akad. Nauk SSSR*. 1977. V. 236. N. 5. P. 1091–1094.
3. *Ryzhov O. S., Savenkov I. V.* On the stability of the boundary layer at transonic external flow velocities // *Prikl. Mekh. Tekhn. Fiz.* 1990. N. 2. P. 65–71.